Table of Contents

Chapter 1: Getting Started ..................................................................................................................21
GS. Welcome to STAAD.Pro ................................................................................................................21
GS. Overview of the STAAD.Pro Environment .................................................................................21
GS. About STAAD.Pro Documentation ..............................................................................................22
   GS. Using Online Help .....................................................................................................................24
   GS. Documentation Conventions .....................................................................................................26
   GS. Where are the old manuals? .....................................................................................................27
GS. About STAAD.Pro .........................................................................................................................28
   GS. System Requirements ................................................................................................................28
   GS. Installation and Licensing .........................................................................................................30
   GS. Limits on Models .......................................................................................................................31
GS. Fundamentals ................................................................................................................................31
   GS. Starting STAAD.Pro ....................................................................................................................31
   GS. To create a new STAAD.Pro model .............................................................................................33
   GS. To open a STAAD.Pro model .......................................................................................................34
   GS. Workflows in STAAD.Pro .........................................................................................................35
   GS. Selecting Objects in STAAD.Pro .................................................................................................37
   GS. Units in STAAD.Pro ...................................................................................................................39
   GS. Coordinates in STAAD.Pro .........................................................................................................44
   GS. Load Types in STAAD.Pro .........................................................................................................47
   GS. STAAD Input Files .....................................................................................................................49
   GS. Object Properties Inspection ....................................................................................................49
GS. Application Window Layout ..........................................................................................................50
   GS. Start Page ..................................................................................................................................50
   GS. Quick Access Toolbar ...............................................................................................................57
   GS. Tool Search .................................................................................................................................58
   GS. Page Control ..............................................................................................................................59
   GS. Data Area ...................................................................................................................................61
   GS. View Window .............................................................................................................................61
   GS. Status Bar ....................................................................................................................................78
GS. Keyboard Shortcuts .......................................................................................................................78
GS. Services and Support Information .................................................................................................81
Nuclear Safety Related Features .........................................................................................................82

Chapter 2: What's New? ........................................................................................................................84
STAAD.Pro CONNECT Edition V22 ......................................................................................................84
   CONNECT Edition V22 Update 1 .......................................................................................................84
   CONNECT Edition V22 ......................................................................................................................90
STAAD.Pro CONNECT Edition ............................................................................................................91
   CONNECT Edition Update 3 ..............................................................................................................91
   CONNECT Edition Update 2 ..............................................................................................................98
   CONNECT Edition Update 1 .............................................................................................................103
   CONNECT Edition ..........................................................................................................................105
STAAD.Pro V8i ......................................................................................................................................108
   V8i (SELECTseries 6) .........................................................................................................................108
Chapter 3: Tutorials ................................................................. 373

T.1 - Steel Portal Frame .......................................................... 373
  T.1 Methods of creating the model ....................................... 373
  T.1 Description of the tutorial problem ............................... 374
  T.1 Creating a new structure ................................................ 375
  T.1 Creating the Model using the Physical Modeler ......... 377
  T.1 Creating the model using the analytical user interface ... 385
  T.1 Analysis and Design ....................................................... 412
  T.1 Viewing the input command file ................................... 427
  T.1 Creating the model using the command file ............... 428
  T.1 Performing Analysis/Design .......................................... 432
  T.1 Viewing the output file ................................................ 433
  T.1 Post-Processing ............................................................. 441

T.2 - RC Framed Structure .................................................. 449
  T.2 Methods of creating the model ..................................... 449
  T.2 Description of the tutorial problem ............................... 449
  T.2 Creating a new structure .............................................. 451
  T.2 Creating the Model using the Physical Modeler ......... 452
  T.2 Creating the model using the analytical user interface ... 458
  T.2 Analysis and Design ....................................................... 458
  T.2 Viewing the input command file ................................... 483
  T.2 Creating the model using the command file ............... 494
  T.2 Performing the analysis and design ......................... 499
  T.2 Viewing the output file ................................................ 501
  T.2 Post-Processing ............................................................. 506

T.3 - Analysis of a slab ....................................................... 535
  T.3 Methods of creating the model ..................................... 535
  T.3 Description of the tutorial problem ............................... 536
  T.3 Creating a new structure .............................................. 536
  T.3 Creating the Model using the Physical Modeler ......... 538
  T.3 Creating the model using the analytical user interface ... 539
  T.3 Analysis and Design ....................................................... 547
  T.3 Post-Processing ............................................................. 578
Chapter 4: Modeling .............................................................. 615
M. Navigating the Graphical View Window ................................................................. 615
M. To select a center of rotation at a node ................................................................. 615
M. To view a 3D rendering of your model ................................................................. 616
M. Labels .................................................................................................................. 617
M. Views .................................................................................................................. 623
M. To insert custom text in a view ............................................................................ 630
M. To display loads graphically .................................................................................. 631
M. To identifying beam start and end ....................................................................... 634
M. Rotation tools ...................................................................................................... 636
M. To display master nodes ...................................................................................... 638
M. Creating Model Objects ...................................................................................... 639
M. Drawing Aids ...................................................................................................... 639
M. Beams .................................................................................................................. 643
M. Physical Members ................................................................................................ 649
M. Plates ................................................................................................................... 653
M. Composite Decks ................................................................................................. 693
M. Solids ................................................................................................................... 696
M. Nodes ................................................................................................................... 697
M. Modify Your Model .............................................................................................. 699
M. Groups ............................................................................................................... 704
M. Structure Wizard .................................................................................................. 710
M. Pages in the Analytical Modeling Workflow ...................................................... 729
M. Properties and Specifications ............................................................................. 730
M. Section Profiles .................................................................................................... 730
M. Materials and Constants ...................................................................................... 788
M. Member Orientation .............................................................................................. 794
M. Member Specifications ......................................................................................... 797
M. Plate Specifications ............................................................................................... 805
M. Node Specifications ............................................................................................... 810
M. Supports ............................................................................................................. 812
M. To assign a fixed or pinned support ..................................................................... 812
M. To assign an enforced support ............................................................................. 813
M. To assign custom release supports ...................................................................... 814
M. To assign a spring support .................................................................................... 814
M. To assign an inclined support .............................................................................. 817
M. To assign a foundation support ............................................................................ 818
M. Loading Your Model ............................................................................................ 819
M. Available Structural Load Specifications in STAAD.Pro ................................. 819
M. To create a new primary load case ...................................................................... 822
M. Load Items .......................................................................................................... 823
M. Wind Loads ........................................................................................................... 835
M. Seismic Loads ...................................................................................................... 840
M. Response Spectra ................................................................................................. 842
M. Snow Loads .......................................................................................................... 851
Chapter 5: Analysis ................................................................. 928

A. Types of Analysis ......................................................................................................................... 928
A. To specify a linear elastic analysis ............................................................................................ 928
A. To specify a P-Delta analysis ...................................................................................................... 928
A. To specify a direct analysis ......................................................................................................... 929
A. To specify a nonlinear analysis .................................................................................................. 930
A. To specify a nonlinear cable analysis ......................................................................................... 931
A. To specify an imperfection analysis .......................................................................................... 932
A. To specify buckling analysis ..................................................................................................... 932
A. To specify a pushover analysis .................................................................................................. 933
A. To add a change command ...................................................................................................... 934
A. To generate a floor spectrum .................................................................................................... 935
A. To specify pre-analysis commands .......................................................................................... 936
A. To create a load list ..................................................................................................................... 937
A. To check for soft stories and seismic code irregularities .......................................................... 937

M. Notional Loads .............................................................................................................................. 852
M. Moving Loads .............................................................................................................................. 853
M. Time History Loads ..................................................................................................................... 855
M. Pushover Loads .......................................................................................................................... 861
M. To use starting vectors with load-depant Ritz vectors .............................................................. 865
M. Load Combinations .................................................................................................................... 865
M. To add a repeat load case .......................................................................................................... 872
M. To create a reference load .......................................................................................................... 873
M. Damping Modeling ...................................................................................................................... 874
M. Mass Modeling .......................................................................................................................... 876
M. To create a load envelope .......................................................................................................... 877
M. Piping workflow ......................................................................................................................... 878
M. Bridge Deck workflow ............................................................................................................. 883

M. Checking Your Model ................................................................................................................. 887
M. To check for multiple structures ............................................................................................... 887
M. To check for warped plates ....................................................................................................... 887
M. To check for and remove duplicate entities ............................................................................. 889
M. To detect and remove zero length members ........................................................................... 890
M. To check for overlapping collinear members ......................................................................... 891
M. To change a beam incidence .................................................................................................... 892
M. To detect and remove orphan nodes ....................................................................................... 892
M. To display the distance between two nodes ........................................................................... 892
M. To display beam lengths ........................................................................................................... 893
M. To check for negative volume solids ...................................................................................... 893
M. Physical Modeling workflow .................................................................................................... 894
M. Using the Physical Modeler ...................................................................................................... 894
M. To drop the associated physical model ................................................................................... 894
M. Building Planner workflow ...................................................................................................... 895
M. To start a STAAD Model in the Building Planner workflow .................................................. 895
M. Plans ........................................................................................................................................... 895
M. Slabs .......................................................................................................................................... 899
M. Columns ................................................................................................................................... 902
M. Beams ...................................................................................................................................... 904
M. Frames ..................................................................................................................................... 911
M. Analysis and Design ................................................................................................................ 923
Chapter 6: Design .............................................................. 944

D. Batch Design versus Interactive Design Workflows ................................................................. 944
D. Steel Design ................................................................................................................................. 944
D. Available Steel Design Codes ................................................................................................... 944
D. Batch Steel Design Operations ................................................................................................. 950
D. Steel Connection Design ........................................................................................................ 953
D. Concrete Design ....................................................................................................................... 1000
D. Available Concrete Design Codes ........................................................................................... 1000
D. Batch Member and Element Design Operations ...................................................................... 1002
D. Interactive Concrete Design ................................................................................................. 1005
D. Advanced Concrete Design .................................................................................................... 1352
D. Advanced Slab Design ........................................................................................................... 1353
D. Foundation Design .................................................................................................................. 1359
D. Aluminium Design .................................................................................................................. 1362
D. Available Aluminum Design Codes ........................................................................................ 1362
D. To specify aluminum design code and parameters ................................................................. 1363
D. To specify aluminum design commands ............................................................................... 1363
D. To generate aluminum take off ............................................................................................. 1364
D. Timber Design ....................................................................................................................... 1364
D. Available Timber Design Codes ............................................................................................ 1364
D. To specify timber design code and parameters ..................................................................... 1365
D. To specify timber design commands ..................................................................................... 1365
D. Design Codes ......................................................................................................................... 1366
D1. American Codes .................................................................................................................... 1366
D2. Australian Codes .................................................................................................................. 1627
D3. British Codes ....................................................................................................................... 1655
D4. Canadian Codes ................................................................................................................... 1695
D5. European Codes ................................................................................................................. 1742
D6. French Codes ........................................................................................................................ 1872
D7. German Codes ..................................................................................................................... 1880
D8. Indian Codes ......................................................................................................................... 1887
D10. Mexican Codes ................................................................................................................... 2002
D11. New Zealand Codes .......................................................................................................... 2021
D12. Norwegian Codes .............................................................................................................. 2046
D13. Russian Codes ................................................................................................................... 2113
D14. South African Codes .......................................................................................................... 2176

Chapter 7: Postprocessing and Reports .......................................................... 2212

P. To view analysis results ........................................................................................................... 2212
P. Postprocessing Workflow ...................................................................................................... 2213
P. To reposition support reaction labels .................................................................................... 2213
P. To create a animated video file from analysis results ............................................................ 2213
P. Nodal Results ......................................................................................................................... 2214
P. Beam Results ......................................................................................................................... 2214
Chapter 9: General Engineering Theory

G.1 Input Generation
G.2 Types of Structures
G.3 Unit Systems
G.4 Coordinate Systems and Structure Geometry
  G.4.1 Global Coordinate System
  G.4.2 Local Coordinate System
  G.4.3 Relationship Between Global and Local Coordinates
G.5 Finite Element Information
  G.5.1 Plate and Shell Elements
  G.5.2 Solid Elements
  G.5.3 Surface Elements (Deprecated)
G.6 Member Properties
  G.6.1 Prismatic Properties
  G.6.2 Built-In Steel Section Libraries
  G.6.3 User-Provided Steel Table
  G.6.4 Tapered Sections
  G.6.5 Assign Command
  G.6.6 Steel Joist and Joist Girders
  G.6.7 Composite Beams and Composite Decks
  G.6.8 Curved Members
G.7 Member and Element Release
G.8 Axial-Only Specifications
  G.8.1 Truss and Tension- or Compression-Only Members
  G.8.2 Cable Members
G.9 Connection Tags
G.11 Member Offsets
G.12 Material Properties
G.13 Supports
  G.9 Tension- and Compression- Only Springs
G.14 Rigid Diaphragms
G.15 Loads
  G.15.1 Joint Loads
  G.15.2 Member Load
  G.15.3 Area, One-way, and Floor Loads
  G.15.4 Fixed End Member Load
  G.15.5 Prestress and Poststress Member Load
  G.15.6 Temperature and Strain Load
  G.15.7 Support Displacement Loads

I. Copy/Paste from Spreadsheets
I. Importing Models
I. To import a DXF file
I. To import a CIS/2 file
I. To export to a DXF file
I. To export to a CIS/2 file
I. To export structure date to AutoPipe
I. To export to a SACS input file
I. Command Line Support
I. Command Line Syntax
I. Copy/Paste from Spreadsheets
<table>
<thead>
<tr>
<th>TR.0</th>
<th>STAAD Commands and Input Instructions .................................................</th>
<th>2404</th>
</tr>
</thead>
<tbody>
<tr>
<td>TR.1</td>
<td>Command Language Conventions ...................................................................</td>
<td>2405</td>
</tr>
<tr>
<td>TR.1.1</td>
<td>Elements of STAAD Commands ....................................................................</td>
<td>2406</td>
</tr>
<tr>
<td>TR.1.2</td>
<td>Command Formats .......................................................................................</td>
<td>2407</td>
</tr>
<tr>
<td>TR.1.3</td>
<td>Listing of Objects by Specification of Global Ranges ...............................</td>
<td>2409</td>
</tr>
<tr>
<td>TR.2</td>
<td>Problem Initiation and Model Title ..........................................................</td>
<td>2409</td>
</tr>
<tr>
<td>TR.3</td>
<td>Unit Specification ......................................................................................</td>
<td>2411</td>
</tr>
<tr>
<td>TR.4</td>
<td>Input/Output Width Specification .............................................................</td>
<td>2413</td>
</tr>
<tr>
<td>TR.5</td>
<td>Set Command Specification ........................................................................</td>
<td>2413</td>
</tr>
<tr>
<td>TR.6</td>
<td>Data Separator ...........................................................................................</td>
<td>2422</td>
</tr>
<tr>
<td>TR.7</td>
<td>Page Control Commands .............................................................................</td>
<td>2423</td>
</tr>
<tr>
<td>TR.7.1</td>
<td>Page New ..................................................................................................</td>
<td>2423</td>
</tr>
<tr>
<td>TR.7.2</td>
<td>Page Length ..............................................................................................</td>
<td>2423</td>
</tr>
<tr>
<td>TR.8</td>
<td>Ignore Specifications ..................................................................................</td>
<td>2423</td>
</tr>
<tr>
<td>TR.9</td>
<td>No Design Specification .............................................................................</td>
<td>2424</td>
</tr>
<tr>
<td>TR.10</td>
<td>Job Information Data ..................................................................................</td>
<td>2424</td>
</tr>
<tr>
<td>TR.11</td>
<td>Joint Coordinates Specification ..................................................................</td>
<td>2425</td>
</tr>
<tr>
<td>TR.12</td>
<td>Member Incidences Specification ................................................................</td>
<td>2428</td>
</tr>
<tr>
<td>TR.13</td>
<td>Plate and Solid Elements ..........................................................................</td>
<td>2430</td>
</tr>
<tr>
<td>TR.13.1</td>
<td>Plate and Shell Element Incidence Specification ....................................</td>
<td>2431</td>
</tr>
<tr>
<td>TR.13.2</td>
<td>Solid Element Incidences Specification ..................................................</td>
<td>2432</td>
</tr>
<tr>
<td>TR.13.3</td>
<td>Surface Entities Specification ....................................................................</td>
<td>2433</td>
</tr>
<tr>
<td>TR.14</td>
<td>Plate Element Mesh Generation ................................................................</td>
<td>2433</td>
</tr>
<tr>
<td>TR.14.1</td>
<td>Parametric Mesh Models ..........................................................................</td>
<td>2434</td>
</tr>
<tr>
<td>TR.14.2</td>
<td>Element Mesh Generation .........................................................................</td>
<td>2435</td>
</tr>
<tr>
<td>TR.15</td>
<td>Redefinition of Joint and Member Numbers ...............................................</td>
<td>2439</td>
</tr>
<tr>
<td>TR.16</td>
<td>Entities as Single Objects .........................................................................</td>
<td>2440</td>
</tr>
<tr>
<td>TR.16.1</td>
<td>Listing of Entities by Specifying Groups ...............................................</td>
<td>2440</td>
</tr>
<tr>
<td>TR.16.2</td>
<td>Physical Members ......................................................................................</td>
<td>2443</td>
</tr>
<tr>
<td>TR.17</td>
<td>Rotation of Structure Geometry ..................................................................</td>
<td>2444</td>
</tr>
<tr>
<td>TR.18</td>
<td>Inactive/Delete Specification ....................................................................</td>
<td>2445</td>
</tr>
<tr>
<td>TR.19</td>
<td>User Steel Table Specification ...................................................................</td>
<td>2446</td>
</tr>
<tr>
<td>G.16.1</td>
<td>Moving Load Generator ............................................................................</td>
<td>2343</td>
</tr>
<tr>
<td>G.16.2</td>
<td>Seismic Load Generator ............................................................................</td>
<td>2343</td>
</tr>
<tr>
<td>G.16.3</td>
<td>Wind Load Generator ................................................................................</td>
<td>2345</td>
</tr>
<tr>
<td>G.16.4</td>
<td>Snow Load ..................................................................................................</td>
<td>2345</td>
</tr>
<tr>
<td>G.17.1</td>
<td>Stiffness Analysis .....................................................................................</td>
<td>2346</td>
</tr>
<tr>
<td>G.17.2</td>
<td>Second Order Analysis ..............................................................................</td>
<td>2349</td>
</tr>
<tr>
<td>G.17.3</td>
<td>Dynamic Analysis ......................................................................................</td>
<td>2362</td>
</tr>
<tr>
<td>G.17.4</td>
<td>Pushover Analysis ......................................................................................</td>
<td>2376</td>
</tr>
<tr>
<td>G.18.1</td>
<td>Secondary Analysis ....................................................................................</td>
<td>2395</td>
</tr>
<tr>
<td>G.19</td>
<td>Multiple Analyses ......................................................................................</td>
<td>2400</td>
</tr>
<tr>
<td>G.20</td>
<td>Steel, Concrete, and Timber Design ..........................................................</td>
<td>2402</td>
</tr>
<tr>
<td>G.21</td>
<td>Printing Facilities .......................................................................................</td>
<td>2402</td>
</tr>
<tr>
<td>G.22</td>
<td>Miscellaneous Facilities .............................................................................</td>
<td>2402</td>
</tr>
</tbody>
</table>
TR.19.1 Wide Flange .................................................................2449
TR.19.2 Channel ......................................................................2451
TR.19.3 Angle ........................................................................2452
TR.19.4 Double Angle ..............................................................2452
TR.19.5 Tee ............................................................................2453
TR.19.6 Pipe ...........................................................................2454
TR.19.7 Tube ...........................................................................2454
TR.19.8 General ........................................................................2455
TR.19.9 I Section .................................................................2456
TR.19.10 Prismatic .................................................................2457
TR.19.11 Using Reference Table Files ................................................2458
TR.20 Member Property Specification .................................................................2459
TR.20.0 Assigning Properties from Steel Tables .........................................................2460
TR.20.1 Prismatic Property Specification ...............................................................2461
TR.20.2 Tapered Member Specification .................................................................2462
TR.20.3 Property Specification from User Provided Table ...........................................2463
TR.20.4 Assign Profile Specification ..........................................................2464
TR.20.5 Examples of Member Property Specification .................................................2465
TR.20.6 Composite Sheets .................................................................2466
TR.20.7 Composite Decks .................................................................2467
TR.20.8 Curved Member Specification .................................................................2468
TR.20.9 Applying Fireproofing on members .................................................................2469
TR.20.10 Member Property Reduction Factors ..........................................................2470
TR.21 Element/Surface Property Specification .........................................................2471
TR.21.1 Element Property Specification ...............................................................2472
TR.21.2 Surface Property Specification .........................................................2473
TR.22 Member and Element Releases .................................................................2474
TR.22.1 Member Release Specification ...............................................................2475
TR.22.2 Element Release Specification ...............................................................2476
TR.22.3 Element Ignore Stiffness ..........................................................2477
TR.23 Axial Member Specifications .................................................................2478
TR.23.1 Member Truss Specification ...............................................................2479
TR.23.2 Member Cable Specification ...............................................................2480
TR.23.3 Member Tension/Compression Specification ..........................................2481
TR.24 Element Plane Stress and Ignore Inplane Rotation Specification ..................................................2482
TR.25 Member Offset Specification .................................................................2483
TR.26 Specifying and Assigning Material Constants .................................................................2484
TR.26.1 Define Material .................................................................2485
TR.26.2 Specifying Constants for Members and Elements ...............................................2486
TR.26.3 Surface Constants Specification ...............................................................2487
TR.26.4 Modal Damping Information .................................................................2488
TR.26.5 Composite Damping for Springs .................................................................2489
TR.26.6 Member Imperfection Information ...............................................................2490
TR.27 Support Specifications .................................................................2491
TR.27.1 Global Support Specification ...............................................................2492
TR.27.2 Inclined Support Specification ...............................................................2493
TR.27.3 Automatic Spring Support Generator for Foundations ........................................2494
TR.27.4 Multilinear Spring Support Specification ......................................................2495
TR.27.5 Spring Tension/Compression Specification ..................................................2496
TR.28 Rigid Diaphragm Modeling .................................................................2497
TR.28.1 Master/Slave Specification .................................................................2498
TR.28.2 Floor Diaphragm .................................................................2499
Appendix A: Ribbon Control Reference

File tab
Info tab
New tab
Open tab
Save / Save As tabs
Print tab
Report tab
ISM tab
Import/Export tab
Cloud Services tab
Settings tab
Tools tab
Help tab
Geometry tab
Paste with Move dialog
Translational Repeat dialog
3D Circular dialog
Rotate dialog
Snap Node-Beam dialog
Mirror dialog
Move Entities dialog

TR.41 Section Specification
TR.42 Print Specifications
TR.43 Stress/Force Output Printing for Surface Entities
TR.44 Printing Section Displacements for Members
TR.45 Printing the Force Envelope
TR.46 Post Analysis Printer Plot Specifications
TR.47 Size Specification
TR.48 Steel and Aluminum Design Specifications
TR.50 Group Specification
TR.51 Steel and Aluminum Take Off Specification
TR.52 Timber Design Specifications
   TR.52.1 Timber Design Parameter Specifications
   TR.52.2 Code Checking Specification
   TR.52.3 Member Selection Specification
TR.53 Concrete Design Specifications
   TR.53.1 Design Initiation
   TR.53.2 Concrete Design-Parameter Specification
   TR.53.3 Concrete Design Command
   TR.53.4 Concrete Take Off Command
   TR.53.5 Concrete Design Terminator
TR.54 Footing Design Specifications
TR.55 Shear Wall Design
TR.56 End Run Specification
Index of Commands
<table>
<thead>
<tr>
<th>Section</th>
<th>Dialog</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Move Origin dialog</td>
<td>........................................................................................................</td>
<td>2903</td>
</tr>
<tr>
<td>Select Node dialog</td>
<td>........................................................................................................</td>
<td>2904</td>
</tr>
<tr>
<td>Renumber dialog</td>
<td>........................................................................................................</td>
<td>2904</td>
</tr>
<tr>
<td>Define Member Attributes dialog</td>
<td>........................................................................................................</td>
<td>2905</td>
</tr>
<tr>
<td>Insert Nodes into Beam # dialog</td>
<td>........................................................................................................</td>
<td>2906</td>
</tr>
<tr>
<td>Insert Node / Nodes dialog</td>
<td>........................................................................................................</td>
<td>2907</td>
</tr>
<tr>
<td>Stretch Member(s) dialog</td>
<td>........................................................................................................</td>
<td>2908</td>
</tr>
<tr>
<td>Merge Selected Beams dialog</td>
<td>........................................................................................................</td>
<td>2909</td>
</tr>
<tr>
<td>Parametric Models dialog</td>
<td>........................................................................................................</td>
<td>2910</td>
</tr>
<tr>
<td>Define Meshing Region dialog</td>
<td>........................................................................................................</td>
<td>2914</td>
</tr>
<tr>
<td>Select Meshing Parameters dialog</td>
<td>........................................................................................................</td>
<td>2915</td>
</tr>
<tr>
<td>Define Plate Object Property dialog</td>
<td>........................................................................................................</td>
<td>2916</td>
</tr>
<tr>
<td>Composite Deck dialog</td>
<td>........................................................................................................</td>
<td>2916</td>
</tr>
<tr>
<td>View tab</td>
<td>........................................................................................................</td>
<td>2918</td>
</tr>
<tr>
<td>Diagrams dialog</td>
<td>........................................................................................................</td>
<td>2927</td>
</tr>
<tr>
<td>Orientation dialog</td>
<td>........................................................................................................</td>
<td>2936</td>
</tr>
<tr>
<td>Tables dialog</td>
<td>........................................................................................................</td>
<td>2937</td>
</tr>
<tr>
<td>Open View dialog</td>
<td>........................................................................................................</td>
<td>2938</td>
</tr>
<tr>
<td>Options dialog</td>
<td>........................................................................................................</td>
<td>2938</td>
</tr>
<tr>
<td>Color Manager dialog</td>
<td>........................................................................................................</td>
<td>2942</td>
</tr>
<tr>
<td>Tool Tip Options dialog</td>
<td>........................................................................................................</td>
<td>2944</td>
</tr>
<tr>
<td>Select tab</td>
<td>........................................................................................................</td>
<td>2945</td>
</tr>
<tr>
<td>Visual Check dialog</td>
<td>........................................................................................................</td>
<td>2952</td>
</tr>
<tr>
<td>Select Nodes dialog</td>
<td>........................................................................................................</td>
<td>2953</td>
</tr>
<tr>
<td>Select Groups dialog</td>
<td>........................................................................................................</td>
<td>2953</td>
</tr>
<tr>
<td>Specification tab</td>
<td>........................................................................................................</td>
<td>2954</td>
</tr>
<tr>
<td>Specifications - Whole Structure dialog</td>
<td>........................................................................................................</td>
<td>2959</td>
</tr>
<tr>
<td>Properties - Whole Structure dialog</td>
<td>........................................................................................................</td>
<td>2968</td>
</tr>
<tr>
<td>Property dialog</td>
<td>........................................................................................................</td>
<td>2970</td>
</tr>
<tr>
<td>Property: Tapered I dialog</td>
<td>........................................................................................................</td>
<td>2972</td>
</tr>
<tr>
<td>User Provided Table dialog</td>
<td>........................................................................................................</td>
<td>2973</td>
</tr>
<tr>
<td>Create User Provided Table dialog</td>
<td>........................................................................................................</td>
<td>2974</td>
</tr>
<tr>
<td>Material Constant dialog</td>
<td>........................................................................................................</td>
<td>2977</td>
</tr>
<tr>
<td>Beta Angle dialog</td>
<td>........................................................................................................</td>
<td>2978</td>
</tr>
<tr>
<td>Reference Point dialog</td>
<td>........................................................................................................</td>
<td>2979</td>
</tr>
<tr>
<td>Plate ElementProperty dialog</td>
<td>........................................................................................................</td>
<td>2981</td>
</tr>
<tr>
<td>Plate Reference Point dialog</td>
<td>........................................................................................................</td>
<td>2981</td>
</tr>
<tr>
<td>Loading tab</td>
<td>........................................................................................................</td>
<td>2992</td>
</tr>
<tr>
<td>Load &amp; Definition dialog</td>
<td>........................................................................................................</td>
<td>2994</td>
</tr>
<tr>
<td>Create Primary Load Case dialog</td>
<td>........................................................................................................</td>
<td>3002</td>
</tr>
<tr>
<td>Define Load Combinations dialog</td>
<td>........................................................................................................</td>
<td>3003</td>
</tr>
<tr>
<td>Add New Reference Load Definitions dialog</td>
<td>........................................................................................................</td>
<td>3006</td>
</tr>
<tr>
<td>Add New Load Items dialog</td>
<td>........................................................................................................</td>
<td>3006</td>
</tr>
<tr>
<td>Load Generation dialog</td>
<td>........................................................................................................</td>
<td>3031</td>
</tr>
<tr>
<td>Define Load Type dialog</td>
<td>........................................................................................................</td>
<td>3032</td>
</tr>
<tr>
<td>Auto Load Combination dialog</td>
<td>........................................................................................................</td>
<td>3033</td>
</tr>
<tr>
<td>Edit Load Rules for Auto Load Combination Generator dialog</td>
<td>........................................................................................................</td>
<td>3035</td>
</tr>
</tbody>
</table>
Create Wind Type Definition dialog ................................................................. 3038
Add New Snow Definition dialog ........................................................................ 3043
Add New Seismic Definitions dialog ..................................................................... 3044
Add New Direct Analysis Definition dialog ......................................................... 3047
Add New Vehicle Definitions dialog ..................................................................... 3048
Add New Pushover dialog .................................................................................... 3049
Add New Time History Definitions dialog .......................................................... 3056
Define (Time History) Parameters dialog ............................................................. 3058
Modal Damping dialog .......................................................................................... 3059
Analysis and Design tab ....................................................................................... 3060
Analysis - Whole Structure dialog ....................................................................... 3064
Analysis/Print Commands ...................................................................................... 3065
Analysis/Print Commands dialog (Pre Print) ......................................................... 3075
Analysis/Print Commands dialog (Post Print) ....................................................... 3076
Floor Diaphragm Options dialog ........................................................................... 3079
Load List dialog .................................................................................................... 3080
STAAD Analysis and Design dialog ....................................................................... 3080
Steel Design - Whole Structure dialog ................................................................... 3081
Concrete Design - Whole Structure dialog .......................................................... 3085
Aluminum Design - Whole Structure dialog ........................................................ 3086
Timber Design - Whole Structure dialog ............................................................... 3088
Utilities tab ........................................................................................................... 3089
Improper Connectivity dialog ............................................................................... 3096
List of Duplicate Beams / Plates dialog ................................................................. 3096
Overlapping Plates dialog ..................................................................................... 3097
Display/Remove Dimensions dialog .................................................................... 3097
Section dialog ....................................................................................................... 3097
Create Group dialog ............................................................................................. 3099
Check Connection Tags dialog ............................................................................ 3100
STAAD.Pro Calculator utility ................................................................................ 3101
Create AVI File dialog .......................................................................................... 3101
Macro dialog ......................................................................................................... 3103
Customize User Defined Tools dialog ................................................................. 3106
Export STAAD Model to SACS ............................................................................ 3107
Export STAAD Model to AutoPIPE ....................................................................... 3107
Piping tab .............................................................................................................. 3108
Export Revised Model dialog ............................................................................... 3110
Pipe Model dialog ................................................................................................. 3111
Support Connection Wizard ............................................................................... 3113
Pipe Supports table ................................................................................................ 3117
Transfer Pipe Reactions to Structure Model dialog ................................................. 3119
Bridge Deck tab .................................................................................................... 3122
Roadways dialog .................................................................................................. 3123
Select Plates in Deck dialog ................................................................................ 3124
Define Roadway dialog ......................................................................................... 3125
Load Generator Parameters dialog ...................................................................... 3130
Diagrams dialog .................................................................................................... 3136
Vehicle Database dialog ........................................................................................ 3138
Results tab ............................................................................................................. 3140
Annotation dialog ............................................................................................... 3145
Beam Property dialog .......................................................................................... 3147
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>V. Bent Beam Thermal Loading</td>
<td>3241</td>
</tr>
<tr>
<td>V.02 Trusses</td>
<td>3244</td>
</tr>
<tr>
<td>V. Axial Stress on a Truss Model</td>
<td>3244</td>
</tr>
<tr>
<td>V. Axial Force on a Cable</td>
<td>3246</td>
</tr>
<tr>
<td>V. Axial Force in a 2D Plane Frame 1</td>
<td>3248</td>
</tr>
<tr>
<td>V. Axial Forces on a 3D Space Model</td>
<td>3250</td>
</tr>
<tr>
<td>V. Reactions in a 2D Truss Model 1</td>
<td>3253</td>
</tr>
<tr>
<td>V. Reactions in a 2D Truss Model 2</td>
<td>3255</td>
</tr>
<tr>
<td>V. Reactions in a 2D Truss Model 3</td>
<td>3258</td>
</tr>
<tr>
<td>V. Deflections in a 2D Truss Model</td>
<td>3259</td>
</tr>
<tr>
<td>V. Stress in a 2D Truss Model</td>
<td>3262</td>
</tr>
<tr>
<td>V. Axial Forces in a 2D Plane Frame 2</td>
<td>3264</td>
</tr>
<tr>
<td>V. Roof Truss Axial Forces</td>
<td>3266</td>
</tr>
<tr>
<td>V.03 Frames</td>
<td>3269</td>
</tr>
<tr>
<td>V. 2D Portal Reactions 1</td>
<td>3269</td>
</tr>
<tr>
<td>V. 3x2 Plane Frame Moments</td>
<td>3273</td>
</tr>
<tr>
<td>V. Support Reactions for a Simple Frame</td>
<td>3277</td>
</tr>
<tr>
<td>V. 2D Portal Reactions 2</td>
<td>3280</td>
</tr>
<tr>
<td>V. 2D Portal Reactions Sidesway 2</td>
<td>3283</td>
</tr>
<tr>
<td>V. 1x2 Plane Frame Lateral Load</td>
<td>3285</td>
</tr>
<tr>
<td>V. 2D Portal Reactions Sidesway 1</td>
<td>3287</td>
</tr>
<tr>
<td>V. 2 Bay Frame Moments and Shear</td>
<td>3289</td>
</tr>
<tr>
<td>V. 3D Frame Max Forces</td>
<td>3292</td>
</tr>
<tr>
<td>V.04 Plate and Shell Elements</td>
<td>3295</td>
</tr>
<tr>
<td>V. Cantilever Tube Stresses and Deflection</td>
<td>3295</td>
</tr>
<tr>
<td>V. 2D Cantilever Beam End Deflection 1</td>
<td>3309</td>
</tr>
<tr>
<td>V. 2D Cantilever Beam End Deflection 2</td>
<td>3312</td>
</tr>
<tr>
<td>V. 2D Curved Beam Maximum Stress</td>
<td>3315</td>
</tr>
<tr>
<td>V. 2D Triangular Surface with Thermal Load</td>
<td>3319</td>
</tr>
<tr>
<td>V. 2D Circular Surface Displacements and Stresses</td>
<td>3326</td>
</tr>
<tr>
<td>V. Warped Surface Displacements</td>
<td>3332</td>
</tr>
<tr>
<td>V. Curved Roof Displacements and Stresses</td>
<td>3334</td>
</tr>
<tr>
<td>V. Spherical Shell Displacements</td>
<td>3341</td>
</tr>
<tr>
<td>V. 2D Circular Plate In-Plane Stresses</td>
<td>3344</td>
</tr>
<tr>
<td>V. 2D Rectangular Plate with fixed edges</td>
<td>3350</td>
</tr>
<tr>
<td>V. 2D Plate Thermal Moment and Stress</td>
<td>3355</td>
</tr>
<tr>
<td>V. 2D Surface Displacements</td>
<td>3357</td>
</tr>
<tr>
<td>V. 2D Tapered Beam In-Plane Stress</td>
<td>3361</td>
</tr>
<tr>
<td>V. 2D Surface with Hole Edge Stress</td>
<td>3364</td>
</tr>
<tr>
<td>V. 2D Circular Surface Edge Stress</td>
<td>3370</td>
</tr>
<tr>
<td>V. 2D Retaining Wall</td>
<td>3378</td>
</tr>
<tr>
<td>V. Thermal Load on a Plate</td>
<td>3388</td>
</tr>
<tr>
<td>V.05 Solids</td>
<td>3398</td>
</tr>
<tr>
<td>V. Cantilever Beam End Displacement 1</td>
<td>3398</td>
</tr>
<tr>
<td>V. Cantilever Beam End Displacement 2</td>
<td>3407</td>
</tr>
<tr>
<td>V.06 Loading</td>
<td>3422</td>
</tr>
<tr>
<td>V. IBC / ASCE 7</td>
<td>3422</td>
</tr>
<tr>
<td>V. IS 1893</td>
<td>3478</td>
</tr>
<tr>
<td>V. Moving Load</td>
<td>3546</td>
</tr>
<tr>
<td>V. NRC</td>
<td>3553</td>
</tr>
<tr>
<td>V. Wind Load</td>
<td>3574</td>
</tr>
</tbody>
</table>

*STAAD.Pro User Manual*
EX. US-14 P-Delta Analysis of a Frame Under Seismic Loads ................................................................. 4468
EX. US-15 Wind and Floor Load Generation on a Space Frame ................................................................. 4480
EX. US-16 Time History Analysis for Forcing Function and Ground Motion ........................................... 4493
EX. US-17 User-Provided Tables ............................................................................................................... 4500
EX. US-18 Stress Calculation for Plate Elements ......................................................................................... 4509
EX. US-19 Inclined Supports ...................................................................................................................... 4515
EX. US-20 Generating a Structure in Cylindrical Coordinates ................................................................. 4522
EX. US-21 Analysis of a Structure with Tension-Only Members ............................................................. 4526
EX. US-22 Time History Analysis for Sinusoidal Loading ........................................................................ 4533
EX. US-23 Spring Support Generation for a Slab on Grade ................................................................. 4541
EX. US-24 Analysis of a Concrete Block Using Solid Elements ............................................................ 4552
EX. US-25 Analysis of a Structure with Compression-Only Members ..................................................... 4566
EX. US-26 Modeling a Rigid Diaphragm Using Master-Slave ............................................................... 4574
EX. US-27 Modeling Soil Springs for a Slab on Grade ........................................................................... 4580
EX. US-28 Calculation of Modes and Frequencies of a Bridge .............................................................. 4591
EX. US-29 Time History Analysis of a Frame for Seismic Loads .............................................................. 4605
EX. British Design Examples ..................................................................................................................... 4627
EX. UK-1 Plane Frame with Steel Design ................................................................................................. 4627
EX. UK-2 Area Load Generation on Floor Structure ................................................................................ 4645
EX. UK-3 Soil Springs for Portal Frame ..................................................................................................... 4663
EX. UK-4 Inactive Members in a Braced Frame ......................................................................................... 4670
EX. UK-5 Support Settlement on a Portal Frame ...................................................................................... 4683
EX. UK-6 Prestress and Poststress Loading ............................................................................................ 4687
EX. UK-7 Modeling Offset Connections in a Frame ................................................................................ 4693
EX. UK-8 Concrete Design for a Space Frame .......................................................................................... 4698
EX. UK-9 Modeling Slabs and Shear Walls Using Finite Elements ....................................................... 4707
EX. UK-10 Finite Element Model for a Rectangular Tank ....................................................................... 4716
EX. UK-11 Response Spectrum Analysis of a Frame ............................................................................... 4725
EX. UK-12 Moving Load Generation on a Bridge Deck ........................................................................ 4735
EX. UK-13 Section Displacements for a Frame ....................................................................................... 4743
EX. UK-14 P-Delta Analysis of a Frame Under Seismic Loads ............................................................... 4749
EX. UK-15 Wind and Floor Load Generation on a Space Frame ............................................................. 4760
EX. UK-16 Time History Analysis for Forcing Function and Ground Motion ........................................ 4774
EX. UK-17 User-Provided Tables ............................................................................................................. 4781
EX. UK-18 Stress Calculation for Plate Elements ...................................................................................... 4789
EX. UK-19 Inclined Supports .................................................................................................................... 4795
EX. UK-20 Generating a Structure in Cylindrical Coordinates ............................................................... 4803
EX. UK-21 Analysis of a Structure with Tension-Only Members ............................................................ 4806
EX. UK-22 Time History Analysis for Sinusoidal Loading ..................................................................... 4813
EX. UK-23 Spring Support Generation for a Slab on Grade .................................................................. 4822
EX. UK-24 Analysis of a Concrete Block Using Solid Elements ............................................................ 4833
EX. UK-25 Analysis of a Structure with Compression-Only Members .................................................. 4847
EX. UK-26 Modeling a Rigid Diaphragm Using Master-Slave ............................................................. 4855
EX. UK-27 Modeling Soil Springs for a Slab on Grade ........................................................................ 4862
EX. UK-28 Calculation of Modes and Frequencies of a Bridge .............................................................. 4873
EX. UK-29 Time History Analysis of a Frame for Seismic Loads ............................................................ 4887
EX. Modeling Examples .......................................................................................................................... 4909
EX. Meshed Wall-Slab Connection .......................................................................................................... 4909
EX. Building Planner Workflow Example ............................................................................................... 4915
EX. Steel Design Examples ....................................................................................................................... 4925
EX. Connection Design Example ............................................................................................................ 4925
Getting Started

This help file is intended to guide users who are new to STAAD.Pro as well as experienced users who want specific information on the basics of using the program.

Start by reading through this section which contains basic information on getting STAAD.Pro running on your computer.

It is then strongly recommended that you work through all three of the tutorials. These contain step-by-step procedures of modeling, analysis, design, and post processing different structures.

Next, read through the frequently performed tasks to learn how to perform common operations in the program.

GS. Welcome to STAAD.Pro

The ultimate power tool for computerized structural engineering.

STAAD.Pro CONNECT Edition is the most popular structural engineering software product for 3D model generation, analysis and multi-material design. It has an intuitive, user-friendly GUI, visualization tools, powerful analysis and design facilities and seamless integration to several other modeling and design software products. The software is fully compatible with supported Windows operating systems.

For static or dynamic analysis of bridges, containment structures, embedded structures (tunnels and culverts), pipe racks, steel, concrete, aluminum or timber buildings, transmission towers, stadiums or any other simple or complex structure, STAAD.Pro has been the choice of design professionals around the world for their specific analysis needs.

To see the new features in STAAD.Pro, please read the release report, which is also included as the What’s New? section in the online help.

GS. Overview of the STAAD.Pro Environment

Tip: Click on any label in the figure to jump to that section of the help.
Getting Started
GS. About STAAD.Pro Documentation

The documentation for STAAD.Pro consists of the following sections that make up the online help.
All the manuals can be accessed from the Help facilities of STAAD.Pro.

Getting Started (GS.)

This section contains information on the contents of the STAAD.Pro package, computer system requirements, installation process, copy protection issues and a description on how to run the programs in the package.

What's New?

This section contains the software release report for STAAD.Pro contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 06.
Tutorials (T.)

Tutorials that provide detailed and step-by-step explanation on using the programs are also provided.

Modeling (M.)

This section describes how to model structural elements and loads in STAAD.Pro using the Analytical Modeling workflow.

Analysis (A.)

This section of the help describes the analysis methods, data print commands, and how to perform analysis in STAAD.Pro.

Design (D.)

This section describes how to perform the design of structural elements using STAAD.Pro.

This section contains information on the various concrete, steel, timber, and aluminum design codes that are implemented in STAAD.Pro.

Postprocessing and Reports (P.)

This section describes how to review output, perform post-processing tasks, generate reports, and plot from STAAD.Pro.

Data Files and Interoperability (I.)

This section describes how to manually edit STAAD.Pro input files, import and export data, and interact with STAAD.Pro using external applications.

General Engineering Theory (G.)

This section deals with the theory behind the engineering calculations made by the STAAD engine.

Technical Reference of STAAD Commands (TR.)

This section includes an explanation of the commands available in the STAAD command file.

Ribbon Control Reference

This section describes the tools found in the ribbon toolbars.

Verification Examples (V.)

This section includes examples of STAAD.Pro example models along side reference ("textbook") examples or a set of “hand” calculations. This provides you with validation of standard program calculations.
Application Examples (EX.)

This section includes examples of various problems that can be solved using the STAAD engine. The examples represent various structural analysis and design problems commonly encountered by structural engineers. Additional example models installed with the product are described here in brief.

OpenSTAAD (OS.)

This section describes the application programming interface for STAAD.Pro.

GS. Using Online Help

The Web help opens in your default web browser.

The Help window consists of two panes — the navigation pane on the left and the topic pane on the right.

The navigation pane has the following tabs:

- Contents — used for browsing topics.
- Index — index of help content.
- Glossary — glossary of the help content.
- Search — displays the search results.
- Tool Index — index of the tools.

Hypertext links, which appear in color and are underlined when you hover the pointer over them, let you move easily between related topics.

**Note:** In case you are not able to view the Web help properly in Internet Explorer, make sure you turn off the Enable Protected Mode check box in the Security tab of the Internet Options dialog.

**Note:** The text size in the topic pane on the right is controlled by your web browser. To learn how to change the web page’s text size, consult the web browser’s Help file.

To open the Help window

1. Either:

   Click the Help tool in the top, right corner of the application window

   or

   Press <F1>

   The Help window opens and the Contents tab is displayed.

The Help window consists of two panes — the navigation pane on the left and the topic pane on the right. At the top of the topic pane is a topic banner containing additional navigation tools.
To browse topics using the Contents tab

1. On the Contents tab, click the symbol preceding any link to expand its contents.
2. Continue expanding folders until you reach the desired topic.
3. Select a topic to display its content in the topic pane.

To display the next or previous topic according to the topic order shown in the Contents tab

1. Either:
   - View...  
     Select...  
     the next topic  
     click the right arrow in the help window’s control bar
   - the previous topic  
     click the left arrow in the help window’s control bar

To navigate in this manner, it is not necessary to first select the Contents tab.

To use the index to find help content

1. Select the Index tab.
2. Scroll through the index using the scroll bar to find a specific entry.

   Tip: Press <Ctrl+F> to use your web browser’s find feature to locate instances of a word or phrase in the index.

3. Select the desired entry.
   The content that the selected index entry is referencing displays in the topic pane.

To search for text in the help content

1. In the Enter search terms field, type the word or phrase for which you are searching.
2. Either:
   - Click Search
   - or
   - Press <Enter>
   Results of the search display on the Search tab.
3. Click the title of any search result.
4. (Optional) Search for a term within a topic:
   a. Press <Ctrl+F>
      Your browser’s search dialog opens.
   b. Type the word or phrase for which you are searching.
   c. Click Next to find each occurrence of the text in this topic.

Search results vary based on the quality of the search criteria entered in the Search field. The more specific the search criteria, the more narrow the search results. You can improve your search results by improving the
search criteria. For example, a word is considered to be a group of contiguous alphanumeric characters. A phrase is a group of words and their punctuation. A search string is a word or phrase on which you search.

**GS. Documentation Conventions**

A number of typographical conventions are maintained throughout Bentley documentation, which makes it easier to identify and understand the information presented.

**Notes, Hints, and Warnings**

Items of special note are indicated as follows:

- **Note**: This is an item of general importance.
- **Tip**: This is optional time-saving information.
- **Warning**: This is information about actions that should not be performed under normal operating conditions.

**File Path/File Name.extension**

A fixed width typeface is used to indicate file names, file paths, and file extensions (e.g., `.../STAAD/Staadpro.exe`).

**Interface Control**

A bold typeface is used to indicate user controls, such as ribbon tabs, tool names, and dialog controls. (e.g., **File > Save As**).

**User Input**

A bold, fixed width typeface is used to indicate information which must be manually entered. (e.g., Type **DEAD LOAD** as the title for Load Case 1).

### Terminology

- **Click** - This refers to the action of pressing a mouse button. When not specified, click means to press the left mouse button.
- **Select** - Synonymous with Click. Used when referring to an action in a menu, drop-down list, list box, or other control where multiple options are available to you.
- **pop-up menu** - A pop-up menu is displayed typically with a right-click of the mouse on an item in the interface.
- **Window** - Describes an on screen element which may be manipulated independently. Multiple windows may be open and interacted with simultaneously.
- **Dialog** - This is an on screen element which (typically) must be interacted with before returning to the main window.
- **Cursor** - Various selection tools are referred to as “cursors” in STAAD.Pro. Selecting one of these tools will change the mouse pointer icon to reflect the current selection mode.

### Mathematical Notation

Similar to spelling conventions, American mathematical notation is used throughout the documentation. A serif typeface is typically used to clarify numbers or letters which might otherwise appear similar.

- Numbers greater than 999 are written using a comma (,) to separate every three digits.
  
  For example, the U.S. value of Young's Modulus is taken as 29,000,000 psi.

**Warning**: Do not use commas or spaces to separate digits within a number in a STAAD input file.
• Numbers with decimal fractions are written with a period to separate whole and fraction parts. For example, a beam with a length of 21.75 feet.
• Multiplication is represented with a raised – or middle– dot (·) or a multiplication symbol (×). For example, \( P = F \cdot A \) or \( P = F \times A \).
• Operation separators are used in the following order:
  1. parenthesis ( )
  2. square brackets [ ]
  3. curly brackets (i.e., braces) { }

For example,

\[
F_a = [1 - (Kl/r)^2/(2 \cdot C_c^2)]F_y / \{5/3 + [3(Kl/r)/(8 \cdot C_c)] - [(Kl/r)^3/(8 \cdot C_c^3)]\}
\]

Which may also be represented as:

\[
F_a = \frac{1 - \left(\frac{Kl}{r}\right)^2}{2C_c^2} F_y \left\{ \frac{5}{3} + \frac{3\left(\frac{Kl}{r}\right)}{8C_c} - \left(\frac{\left(\frac{Kl}{r}\right)^3}{8C_c^3}\right) \right\}
\]

GS. Where are the old manuals?

For many versions of STAAD.Pro, the help was primarily a collection of the user manuals in an online format. Though the organization has changed somewhat, all of the existing content is still available to you.


This chapters of this document have been separated into the logical portions of this help file. You can find each of the sections as follows:

1. “General Description” - General Engineering Theory (on page 2294)
2. “American Steel Design” - D1. American Codes (on page 1366)
3. “American Concrete Design” - D1.F. American Codes - Concrete Design per ACI 318 (on page 1478)
5. “Commands and Input Instructions” - Technical Reference of STAAD Commands (on page 2404)

International Design Codes Manual

This content is included in the D. Design Codes (on page 1366) section of the online help.

The American codes are now included here for consistency. The sections including American aluminium design, American transmission tower design, API steel design code, ANSI/AISC N690 design, and ASME NF codes have all been moved to the chapter for American design codes as well.

Release Report

This document is included as the What’s New? (on page 84) section of the online help.
Getting Started

This document has been reorganized and is included primarily within the Getting Started section of the online help.

Graphical User Interface

This content is no longer collected into a sub-section, but has been expanded to cover modeling, analysis, design, post-processing, and interoperability as described elsewhere in the online help.

Examples Manual

Most complete input examples, verification examples, and extended modeling examples have been collected into the Application Examples section of the help.

GS. About STAAD.Pro

STAAD.Pro is a general purpose structural analysis and design program with applications primarily in the building industry — commercial buildings, bridges and highway structures, industrial structures, chemical plant structures, dams, retaining walls, turbine foundations, culverts and other embedded structures, etc. The program hence consists of the following facilities to enable this task.

1. Graphical model generation utilities as well as text editor based commands for creating the mathematical model. Beam and column members are represented using lines. Walls, slabs and panel type entities are represented using triangular and quadrilateral finite elements. Solid blocks are represented using brick elements. These utilities allow you to create the geometry, assign properties, orient cross sections as desired, assign materials like steel, concrete, timber, aluminum, specify supports, apply loads explicitly as well as have the program generate loads, design parameters etc.

2. Analysis engines for performing linear elastic and p-delta analysis, finite element analysis, frequency extraction, and dynamic response (spectrum, time history, steady state, etc.).


4. Result viewing, result verification and report generation tools for examining displacement diagrams, bending moment and shear force diagrams, beam, plate and solid stress contours, etc.

5. Peripheral tools for activities like import and export of data from and to other widely accepted formats, links with other popular software programs for niche areas like reinforced and prestressed concrete slab design, footing design, steel connection design, etc.

6. A library of exposed functions called OpenSTAAD which allows you to access the internal functions and routines in STAAD.Pro as well as its graphical commands to tap into STAAD.Pro’s database and link input and output data to third-party software written using languages like C, C++, VB, VBA, FORTRAN, Java, Delphi, etc. Thus, OpenSTAAD can be used to link in-house or third-party applications with STAAD.Pro.

GS. System Requirements

The following hardware requirements are suggested minimums. Systems with increased capacity provide enhanced performance.
### Processor
Intel® Pentium or AMD® processor 3.0 GHz or greater.

### Memory
1 GB minimum, 2 GB recommended (4GB for STAAD.Pro Advanced). More memory almost always improves performance, particularly when working with larger models. 4 GB (8 GB for STAAD.Pro Advanced) or more can help speed up solutions for very large complex models with large numbers of load cases.

### Video
Graphics card supported by OpenGL. See the graphics card manufacturer for latest information on graphics drivers. 256 MB of video RAM or higher is recommended. If insufficient video RAM or no graphics card supported by OpenGL can be found, the application will attempt to use software emulation. For optimal performance, graphics display color depth should be set to 24-bit or higher. When using a color depth setting of 16-bit, some inconsistencies will be noted.

### Screen Resolution
A minimum screen resolution of 1280x1024 is required, but higher is recommended.

### Hard Disk
Requirements will vary depending on the modules you are installing. A typical minimum is 500 MB free space.
**Supported Operating Systems**

- Windows 10 (64-bit) - Home, Pro, Enterprise, and Education
- Windows 8.1 (64-bit) - Standard, Pro, and Enterprise
- Windows 7 SP1 (64-bit) - Home Basic, Home Premium, Professional, Enterprise, and Ultimate

**Note:** Windows 7 operating system is supported only if you have its service pack (SP1) installed.

**Note:** You must have the following Microsoft updates installed on your machine:

- KB4340917 - For Windows 10 Version 1803 Builds prior to 10.0.17134.191. The changes in KB4340917 are built into the August 2018 monthly update of Windows 10.
- KB2999226 - For Windows 7 SP1, Windows 8.1, Windows Server 2008 R2 SP1, and Windows Server 2012
- KB2999226 - For Windows 8.1 (64bit)
- KB2919355 - For Windows 8.1 and Windows Server 2012

Bentley does not support its software running on Microsoft operating systems versions that Microsoft has "retired". For more information on Microsoft's application retirement policy, click [here](#). For similar information on Bentley products, refer to the [Bentley Product Support](#) article.

Additional RAM, disk space, and video memory will enhance the performance of STAAD.Pro.

The minimum amount of physical + virtual memory required by the program is over 600MB. You may need to ensure that adequate amounts of virtual memory are available and that parameters such as paging file sizes should be large enough or span over multiple drives if the free space on any one drive runs low.

Another issue to keep in mind is the location of the TEMP parameter as in the SET TEMP environment variable in Windows. While performing calculations, depending on the structure size, the program may create very large scratch files which are placed in the folder location associated with the TEMP parameter. You may want to point the SET TEMP variable to a folder on a drive that has disk space sufficiently large to accommodate the requirements for large size structures.

You should have a basic familiarity with Microsoft® Windows® systems in order to use the software.

### GS. Installation and Licensing

For details on installation and licensing of this product, please refer to the [product Readme file](#), which is installed in the same location as the product and this help file.
GS. Limits on Models

The following limits to model size are effective for STAAD.Pro CONNECT Edition (release 21.00.00) and later.

- Joint number 1 to 999,999
- Number of joints: 200,000*
- Member/Element numbers: 1 to 999,999
- Number of Members, Plates, and Solids: 225,000*
- Load Case numbers: 1 to 99,999
- Number of primary and combination cases: 10,101
- Number of modes and frequencies: 2,700
- Number of load cases that may be combined by a Repeat Load or Load Combination command: 550

* Some STAAD copies are available with much smaller limits, please check what limits you have purchased.

Notes:

The numerical limits should be considered as upper limits built into the software for those quantities on an individual basis. In practice, the actual maximums the program can handle are determined by the hardware resources as well as the limits imposed by the operating system. For example, it is highly improbable that a single model with 999,999 members and 99,999 load cases can be solved.

The memory demand of the program is determined by the combined effect of two or more of these terms. For example, when a steel design is performed, the memory required depends on the product of the members being designed (NMD) as well as the number of load cases being designed for (NL). That is, NMD × NL. So the smaller the NMD, the larger the NL capacity and vice versa.

Experience has shown that for a medium size model of 5,000 frame members, about 1,500 load cases is the practical limit of what can be run for the 32-bit program on a computer with 16 GB RAM and 200 GB of free disk space.

GS. Fundamentals

This section introduces you to some of the fundamental concepts necessary for using STAAD.Pro. You are strongly encouraged to review this section to gain a firm understanding of the core concepts of the program.

GS. Starting STAAD.Pro

1. Start STAAD.Pro by one of the following methods:
   double-click the STAAD.Pro CONNECT Edition icon on your desktop
or
click the Windows Start button and then select **Bentley Engineering > STAAD.Pro CONNECT Edition V22**
or
in the program group, select the STAAD.Pro icon.
or
in Windows Explorer, double-click a STAAD.Pro file (with the file extension .std)
or
in Windows Explorer, double-click the icon for STAADPro.exe.

**Note:** Until you have activated a license for the program, it will remain in Limited mode and only function for 15 minutes before shutting down the application.

The **STAAD.Pro License Information** dialog opens.

2. (Optional) To configure the license for this session:
   a. Click
      The **STAAD.Pro License Information** dialog opens.
b. Select the **Product Type** and any **Additional Licenses** you need to use for this session.

Please refer to the product ReadMe or click the **Help with license selection** link for additional details.

c. Click **Accept**.

The STAAD.Pro window opens to the Start page.

3. Click **Continue**.

The STAAD.Pro window opens to the Start page.

**Related Links**

- *GS. STAAD.Pro License Configuration dialog* (on page 53)
- *GS. STAAD.Pro License Configuration dialog* (on page 53)

**GS. To create a new STAAD.Pro model**

To create a new STAAD.Pro model, use the following procedure.

1. On the Start page, select **New**.

   The **New** page opens to the **Model Info** tab.

2. Enter a **File Name**.

3. Specify a **Location** where the STAAD input file will be located on your computer or network.

   You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.

4. Select the **Type** of model.

   This option selects the modeling method you want to use:
Analytical  For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.

Physical  For creating a model using the STAAD.Pro Physical Modeler interface.

Building  For creating a building model structure using the Building workflow.

5. Select the system of **Units**.

- **English**  English Imperial units: feet, pounds, seconds
- **Metric**  System International units: meters, kilograms, seconds

**Tip:** The units can be changed later if necessary, at any stage of the model creation.

6. (Optional) Select the Job Info tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.

7. Click **Create**.

The new STAAD.Pro model is opened in either the Analytical Modeling workflow, STAAD.Pro Physical Modeler, or the Building workflow based on your selection.

**Related Links**
- **GS. Start Page** (on page 50)
- **ProjectWise Project Association** (on page 2273)

**GS. To open a STAAD.Pro model**

To open an existing STAAD.Pro model, use the following procedure.

1. On the Start page, select **Open**.
   The New page opens to the **Recent** tab.
   Recently opened models are organized by the following categories:
   - Analytical
   - Physical
   - ProjectWise
   - Archive
   - All

2. Either:
   - select a recently used model from one of these tabs
   - or
   - select **Open Other Models** to search for a STAAD input file
   - or
   - select either **Sample Models** or **Verification Models** to open the example models which are installed with the program
If you select a recent model, it will open. If you select to browse for a model, an **Open** dialog opens.

3. In the Open dialog, navigate to and select a STAAD.Pro file (file extension *.std*) and then click **Open**.

The selected file opens.

If the STAAD.Pro file does not have a ProjectWise Project associated with it, then you will be prompted to select one.

**Related Links**
- *GS. Start Page* (on page 50)
- *ProjectWise Project Association* (on page 2273)

**GS. Workflows in STAAD.Pro**

The program is organized to reflect the typical process of modeling, analyzing, and post-processing for a structure.

A workflow in STAAD.Pro groups all of the common tasks associated with a major stage of your structural project.

**Overview of Workflows**

<table>
<thead>
<tr>
<th>Workflows</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Analytical Modeling</strong></td>
<td>used to model your structure using analytical objects</td>
</tr>
<tr>
<td><strong>Physical Modeling</strong></td>
<td>opens the STAAD.Pro Physical Modeler application, which is used to model using physical objects which will be subsequently decomposed into analytical objects when exported back into the STAAD.Pro interface</td>
</tr>
<tr>
<td><strong>Building Planner</strong></td>
<td>used to generate concrete building models with the integrated STAAD Building Planner interface</td>
</tr>
<tr>
<td><strong>Piping</strong></td>
<td>used to import piping support geometry and reactions from Bentley AutoPIPE</td>
</tr>
<tr>
<td><strong>Bridge Deck</strong></td>
<td>generating loads for use in the analysis of bridge structures</td>
</tr>
<tr>
<td>Workflow</td>
<td>Description</td>
</tr>
<tr>
<td>------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Postprocessing</td>
<td>used to review the results of analysis and design as well as to build reports</td>
</tr>
<tr>
<td>Foundation Design</td>
<td>used to select load and support data to send to STAAD Foundation Advanced for design</td>
</tr>
<tr>
<td>Steel AutoDrafter</td>
<td>used to generate construction drawings for steel-framed structures</td>
</tr>
<tr>
<td>Connection Design</td>
<td>used to specify steel connection types and design them in RAM Connection</td>
</tr>
<tr>
<td>Concrete Design</td>
<td>opens the RC Designer application, which is an interactive concrete design module</td>
</tr>
<tr>
<td>Advanced Concrete Design</td>
<td>opens the Advanced Concrete Design (RCDC) application, which is used for the design of concrete building structures</td>
</tr>
<tr>
<td>Advanced Slab Design</td>
<td>used to define concrete slabs and transfer data to RAM Concept</td>
</tr>
<tr>
<td>Earthquake</td>
<td>this postprocessing workflow is used to check if the structure conforms to the basic geometric recommendations made in Eurocode 8 (EC8).</td>
</tr>
</tbody>
</table>

**Tip:** Once you are familiar with the Workflows, you can free up some screen area by unpinning the Workflows panel from the program window. It will then collapse to a tab on the left side of the application window. You can expand the panel by clicking on the tab to display the workflows when needed.
Where did the Modes and Page Controls go?

In previous generations of STAAD.Pro, a series of page controls were listed vertically along the left side of the view window. A series of operation modes were listed along the top of the view window (and also accessed via the Modes menu).

These modes have now been replaced by the Workflows panel. Similarly, the page controls are available for the current workflow just below the ribbon bar. These are now organized in a single row, rather than the former groupings.

GS. Selecting Objects in STAAD.Pro

With even relatively small structural models containing many nodes, beams, plates, loads, and other model objects, it is important that you be able to select the correct objects for assigning parameters. STAAD.Pro has a variety of tools to allow you to select objects by type, filter, boolean logic, and more.

In certain modes of selection, objects are chosen by clicking on their entry in a list. For selecting more than one object, press and hold <Ctrl> while clicking.

Selection Method

STAAD.Pro allows you to select one of several methods for graphical selection. These are described in the table below.

To choose a different the selection method, either:

- Select it from the Select ribbon tab in the Modes group, or
- right-click in the view window and select Selection Mode from the pop-up menu.
<table>
<thead>
<tr>
<th>Selection Type</th>
<th>Description</th>
<th>Mouse Action</th>
<th>Resulting Selection Set</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag box</td>
<td>A rectangular selection box activated by holding down the left mouse button and dragging the pointer to create a windowed area in the View window.</td>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
<td>&lt;Ctrl+Shift+F3&gt;</td>
</tr>
<tr>
<td>Tip: This is sometimes referred to as a “rubber band” window.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Drag line</td>
<td>Click and hold down the left mouse button to draw a line. All entities which the line passes through will be selected.</td>
<td><img src="image3.png" alt="Image" /></td>
<td><img src="image4.png" alt="Image" /></td>
<td>&lt;Ctrl+Shift+F3&gt;</td>
</tr>
</tbody>
</table>
### GS. Units in STAAD.Pro

STAAD.Pro uses a set of base units, called current units, for all input and output. The Current Units can be changed at any time to simplify entering input or interpreting results. The current input units are set using the **Set Current Input Units** dialog.

**Tip:** There are several instances in the program where the Current Units cannot be changed, such as when an input dialog is open. In this case, you can use the `<F2>` key to input different units.

STAAD.Pro also includes an extensive unit conversion utility.

### Display Units

STAAD.Pro can use a different set of units for displaying values in the active view window. These are controlled through several tabs on the **Options** dialog.

### Base Unit System

There are two base unit systems in the program which control the units (i.e., length, force, temperature, etc.) in which, values, specifically results and other information presented in the tables and reports, are displayed in. The base unit system also dictates what type of default values the program will use when attributes such as Modulus of Elasticity, Density, etc., are assigned based on material types - Steel, Concrete, Aluminum - selected from the program’s library (Please refer to [Technical Reference of STAAD Commands](on page 2404) for details). These two unit systems are:

- English (Foot, Pound, etc.) and
- Metric (KN, Meter, etc.)

**Tip:** If you recall, one of the choices made at the time of installing STAAD.Pro is this base unit system setting. That choice will serve as the default until you specifically change it.
Selecting Different Input Units in Dialogs

While you may typically wish to work in one unit system - or even with a specific unit of force or length, it is not uncommon to use a different unit for input. STAAD.Pro allows for the input of a value in any unit in most dialog boxes. This avoids the need to manually change the unit before and after adding a command or to convert a unit to something less commonly used. A unit is entered in the most convenient format and then converted to the current units for you. This is especially useful when adding or editing loads, where the current input units cannot be changed.

GS. To change the system units

There are two base unit systems in the program which control the units (length, force, temperature, etc.) in which values — specifically results and other information presented in the tables and report — are displayed. The base unit system also dictates what type of default values the program uses when attributes such as Modulus of Elasticity, Density, etc., are assigned based on material types (i.e., steel, Concrete, Aluminum) selected from the program’s library (Please refer to Built-In Material Constants (on page 2506)). These two unit systems are English (Foot, Pound, etc.) and Metric (KN, Meter, etc.).

1. On the Start page, click Configure.

The Configure Program dialog opens.

2. Select appropriate system of units in the Base Unit group on the General.
3. Click OK.

Related Links

- TR.26.2 Specifying Constants for Members and Elements (on page 2503)
- GS. Application Configuration dialog (on page 55)

GS. To set the current input units

To change the current length and force units for inputs, use the following procedure.

The current input units used for creating the model and assigning attributes such as properties, offsets, loads, etc. can be changed to use convenient units.

1. On the Geometry ribbon tab, select the Input Units tool in the Structure group.

The Set Input Units pop-up opens.
2. Select the base system of **Input Units**:

   - **English**
   - or
   - **Metric**
   - or
   - **Custom** - to mix different systems of units

3. Select a unit for **Length** on the left and unit for **Force** on the right.

4. Click **Apply**.

**GS. To convert units within a dialog field**

To input a unit different than the current unit within a dialog field, use the following procedure.

1. Click within a field that takes a dimensional value as input.

2. Press `<F2>`.

   A pop-up dialog opens to allow you to input values in any dimensionally correct unit.

3. (Optional) Press `<Ctrl+Q>` to open a list of dimensionally correct units and select one to populate this pop-up dialog.

4. Type the value in the known units.

   **Tip:** Values can also be input using a fractional system (e.g., 5 7/8”).

5. Once you are finished entering the units, press `<Return>`. The unit is converted into the Current Unit system.
GS. STAAD.Pro Converter utility

An extensive unit converter is included with the program which can convert values from one unit system to another.

Opens when the Unit Converter tool is selected from the Tools group on the Utilities ribbon tab.

Tools

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image.png" alt="Custom" /></td>
<td>Click to add a custom unit conversion to the current unit category. In the Custom Units table, type a From unit label, the conversion Factor, and a To unit label and then click Ok. For example, you can type the following to add yards to the Length conversion:</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>From</th>
<th>Factor</th>
<th>To</th>
</tr>
</thead>
<tbody>
<tr>
<td>Feet</td>
<td>3</td>
<td>Yards</td>
</tr>
</tbody>
</table>
### Settings dialog

Click the **Settings** tool to display the **Unit Converter** settings.

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Theme</strong></td>
<td>Select a display theme from the drop-down list.</td>
</tr>
<tr>
<td><strong>Output decimal places</strong></td>
<td>Set the decimal places to display in the conversion output.</td>
</tr>
<tr>
<td><strong>Max no of favourite items</strong></td>
<td>Set the number of items to display in the favourites list.</td>
</tr>
<tr>
<td><strong>Max no of recently used items</strong></td>
<td>Set the number of items to display in the recently used items list.</td>
</tr>
<tr>
<td><strong>Top Most</strong></td>
<td>Check this settings to keep the <strong>Unit Converter</strong> dialog on top of all other windows on your screen, even when other windows are active.</td>
</tr>
</tbody>
</table>

### GS. Coordinates in STAAD.Pro

STAAD.Pro Physical Modeler uses a conventional Cartesian coordinate system, with the global Y axis assumed as vertical (i.e., "Y up", or the height of the structure is parallel to the global Y axis).

**Global Coordinate System**

This coordinate system is a rectangular coordinate system (X, Y, Z) which follows the orthogonal "right hand" rule. This coordinate system may be used to define the joint locations and loading directions. The translational degrees of freedom are denoted by u1, u2, & u3 and the rotational degrees of freedom are denoted by u4, u5, & u6.
Local Coordinates

Each member or surface has its own local Cartesian coordinate system which is also oriented using the “right hand” rule.

The longitudinal axis of a member is the first axis, with the positive axis taken from the i to the j ends (i.e., Ni and Nj). The second axis is the oriented such that the 1-2 plane is then parallel to the global Y axis. The third axis is then normal to the first and second local axes as defined by the right hand rule. The local 2 and 3 axes coincide with the two principle moments of inertia of the cross-section. In the special case of a vertical member (where the local 1 axis is parallel to the global Y axis; i.e., a column), the local 3 axis is then made parallel to the global Z axis and the 2 axis is oriented respectively.

For surfaces, the local x axis is aligned with the edge defined by the first two nodes of the surface (i.e., N1 and N2). The program then calculates the area of each triangle formed by any other nodes to determine the largest
area. This triangle determines the plane of the surface and the local y axis lies perpendicular to the x axis within this plane. The third axis is then orthogonal to the surface as defined by the “right hand” rule.

![Figure 3: Local coordinates of a surface](image)

**GS. To use Z as the vertical axis**

To re-orient the global axis so that Z is vertical, use the following procedure.

You must close any open STAAD.Pro models in order change program configuration.

![Y as vertical](image)  ![Z as vertical](image)

**Note:** Both options follow the “right hand rule.”

*Figure 4: Global axis orientation options in STAAD.Pro*

**Important:** Not all STAAD.Pro features are compatible with Z UP orientation. Refer to [SET { Y | Z } UP](on page 2418) for additional details.

1. On the Start page, select **Configure**.
   The Application Configuration dialog opens.
2. On the **General** tab, select **Z up** in the Global Axis group.
3. Click **OK**.
New models will use the Z up orientation of the global axis.

**Tip:** You can repeat this procedure except to select Y up to return to set Y as vertical.

Any input files which have the `SET Z UP` command will automatically use this orientation.

**GS. Load Types in STAAD.Pro**

Several load cases may be created for a structure and each load case may contain several individual load specifications. Load cases may also be created by combining several existing load cases. A load case consisting of explicitly defined loads is called a Primary Load Case. A load case which combines the results of previously defined cases is called a Combination Load Case.

**Tip:** Here, we wish to introduce you to some of the load terminology and types used in STAAD.Pro. These concepts are critical in understanding how to correctly model loads on a structure.

**Primary Load Cases**

A primary load case is a set of explicitly defined loads, presumably from the same physical source, which will be passed to the analysis engine during the analysis of the model. Each of these explicitly defined loads is referred to as a Load Item.

In the STAAD input file, a primary load case is indicated by the `LOAD n` command.

Refer to [TR.32 Loading Specifications](on page 2650) for details.

Some examples of where a primary load case would be used are:

- All dead load on a structure.
- The reducible live load on an office building.
- The easterly wind load.
- The dead and live composite load on a pedestrian bridge.
- The superimposed dead load on a post-tensioned floor.

**Load Combinations**

A load combination is a set of load results which are combined algebraically to produce a superimposed set of results for post-processing. Therefore, a load combination instructs the program to take the results of previously solved primary load cases, factor them appropriately, and combine the values using Algebraic, SRSS or Absolute methods.

In the STAAD input file, a load combination is indicated by the `LOAD COMBINATION n` command.

Refer to [TR.35 Load Combination Specification](on page 2791) for details.
Reference Loads

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. Reference Loads may be defined for this purpose. This is similar to a Repeat Load command, but has the added benefit of not being solved in its own right.

This converts a real load case to something similar to a load case definition. A reference load case is solved only when it is later called in a load case. The benefit is that it enables you to define as many load cases as you wish, but instruct the program to actually solve only a limited number of “real” load cases, thus limiting the amount of results to be examined.

Refer to TR.33 Reference Load Cases - Application (on page 2789) for details.

Notional Loads

A number of design codes require that a notional load be considered. Typically, this is defined lateral load equal to a percentage of the gravity loads. STAAD.Pro uses a feature similar to a the Repeat Load, but one that Primary and /or Reference load cases can be selected and a percentage of them can be applied in the appropriate global direction at each framing level.

These lateral loads are a requirement for some design codes (e.g., AISC 360-10).

Refer to TR.32.14 Notional Loads (on page 2785) for details.

Load Lists

A load list is primarily used to specify a list of existing load cases and load combinations to be used for subsequent processes, such as design, printing, etc.

Refer to TR.39 Load List Specification (on page 2836) for details.

Load Envelopes

Load Envelopes are a means for clustering a set of load cases under a single moniker (number). If one or more tasks have to be performed for a set of load cases (such as, serviceability checks under steel design for one set of load cases, strength checks under steel design for another set of cases, etc.) this feature is convenient.

This is an alternative to Load Lists, and is primarily used in post-processing and design. Load Envelopes also have keyword types which identify their intended use in design.

Refer to TR.40 Load Envelope (on page 2837) for details.

Load Definitions

Definitions contain the options you use to define data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases, snow load cases, and time history load cases.

Refer to TR.31 Definition of Load Systems (on page 2540) for details.
GS. STAAD Input Files

A STAAD model consists of a data file made up of simple, English-language like commands, using a format native to STAAD.Pro. This file specifies all input, analysis, and output commands used to inform the STAAD engine as to the geometry of the structural model, loads acting on this model, and instructs the engine on the analysis and design operations to be performed.

There are two methods for building a model and assigning the structure data using STAAD.Pro:

a. using the graphical model generation, or
b. directly editing the command input file using the STAAD.Pro Editor

The graphical method of creation (a) involves using either the Analytical Modeling or Physical Modeling workflow to draw the model using the graphical tools, and assigning data such as properties, material constants, loads, etc., using the various tools and dialog boxes of that mode. In this approach, the command file is automatically created by the program while you work. The input command file is updated whenever your structure is saved.

The direct editing method (b) involves creating or editing the command input file (file extension .std) directly using the STAAD Editor utility. These commands are explained in detail in Technical Reference of STAAD Commands (on page 2404) and several examples which illustrate this method are provided in the Application Examples (on page 4309).

**Tip:** The command input file is a plain text file so you can use any plain text editing software you prefer. However, it is recommended you use the STAAD.Pro Editor as this program contains tools such as Intellisense to complete commands and syntax checking.

**Tip:** For a hands on illustration of both these methods, it is recommend that you take a look at the Tutorials (on page 373).

Regardless of the means used to create the STAAD input file, once the model is ready for analysis, you will use the graphical environment to initiate analysis, review the analysis and/or design output data, and perform advanced design operations.

GS. Object Properties Inspection

You can inspect the properties and results of most analytical and physical model objects by double-clicking them in the view window.

The dialog that opens reflects the type of object selected as well as the assigned specifications, properties, available analysis and design results, etc.

**Related Links**
- Node dialog (on page 3180)
- Beam dialog (on page 3187)
- Physical Member dialog (on page 3190)
- Plate dialog (on page 3191)
- Surface Query dialog (on page 3192)
- Solid dialog (on page 3194)
GS. Application Window Layout

It is helpful to take some time to familiarize yourself with the components of the window.

1. **File tab** (on page 2869)
2. **Ribbon Control Reference** (on page 2869)
3. **GS. Workflows in STAAD.Pro** (on page 35)
4. **GS. Quick Access Toolbar** (on page 57)
5. **GS. Tool Search** (on page 58)
6. **Bentley CONNECT Features** (on page 2273)
7. **GS. Page Control** (on page 59)
8. **GS. Data Area** (on page 61)
9. **GS. Status Bar** (on page 78)
10. **GS. View Window** (on page 61)

**GS. Start Page**

The start page is displayed when STAAD.Pro opens or there is no model currently open.
Open tab

Used to open an existing STAAD input file or Archive. You can also open STAAD projects from a ProjectWise data source.

Select the Recent tab to view recent files by type. Hover your mouse pointer over any file to display the file and CONNECT Project properties.

The Additional License options are available here to allow you to select which license options you want to use before opening a file.

**Tip:** If you hover your mouse pointer over any item in the recent files list, you will see several tool icons on the right-hand side:

- **Pint to list** – pin this project to the top of the list
- **Remove from list** – remove this project from the list
- **Open Containing Folder** – open the folder where the project is saved in Windows Explorer

New tab

Used to create a new STAAD project. Options are available to start with an analytical model, a physical model, or in STAAD Building Planner.

The Additional License options are available here to allow you to select which license options you want to use before creating a file.

Archive tab

Used to create or extract STAAD.Pro archive files.
Share tab

Table 1: Share menu items

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Send Mail</td>
<td>Used to share STAAD.Pro project files as an e-mail attachment. The selected project files associated with a STAAD.Pro input file are saved as a STAAD.Pro ZIP file (file extension .stz).</td>
</tr>
<tr>
<td>ProjectWise</td>
<td>Used to save STAAD.Pro project files to a ProjectWise data source. The selected project files associated with a STAAD.Pro input file are saved as a STAAD.Pro ZIP file (file extension .stz).</td>
</tr>
</tbody>
</table>

Help tab

Table 2: Help menu items

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td>Opens the online help in your web browser.</td>
</tr>
<tr>
<td>OpenSTAAD Help</td>
<td>Opens the online help in your web browser to the OpenSTAAD reference section.</td>
</tr>
<tr>
<td>Technical Support</td>
<td>Opens the Worldwide Technical Support Resources dialog, which offers a map of worldwide technical support contacts for STAAD.Pro.</td>
</tr>
<tr>
<td>ReadMe</td>
<td>Opens the STAAD.Pro Read Me file in your web browser.</td>
</tr>
<tr>
<td>Knowledge Base and FAQs</td>
<td>Opens the STAAD.Pro Support Solutions page on Bentley Communities in your web browser.</td>
</tr>
<tr>
<td>Discussion Group</td>
<td>Opens the RAM</td>
</tr>
<tr>
<td>Tutorials</td>
<td>Opens the Bentley LEARNserver in your web browser. Here you can access training and learning paths for STAAD.Pro and other Bentley products.</td>
</tr>
</tbody>
</table>
Table 3: About menu items

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>About STAAD.Pro</td>
<td>Opens the About STAAD.Pro CONNECT Edition dialog, which contains version, licensing, and legal information about the product. Any Technical Preview items contained in the current version will also be listed here.</td>
</tr>
<tr>
<td>Product News</td>
<td>Opens the STAAD.Pro product page at Bentley.com in your web browser.</td>
</tr>
<tr>
<td>Home Page</td>
<td>Opens the Bentley.com home page in your web browser.</td>
</tr>
</tbody>
</table>

Related Links

- [GS. To create a new STAAD.Pro model](on page 33)
- [GS. To open a STAAD.Pro model](on page 34)
- [M. To start a STAAD Model in the Building Planner workflow](on page 895)
- [I. To open a STAAD input file from a ProjectWise repository](on page 2284)
- [I. To create an archive](on page 2272)
- [I. To open an archive file](on page 2272)
- [I. To extract an archive](on page 2273)
- [I. To share a STAAD.Pro project in ProjectWise](on page 2285)

GS. **STAAD.Pro License Configuration** dialog

Used to manage license options for STAAD.Pro.

Opens when you select **License** from the Start page menu.
**Product Type**
Select the primary license to use for the application here.

- **STAAD.Pro** – using this license you will have access to the primary features and capabilities available in STAAD.Pro, including all design codes
- **STAAD.Pro Advanced** – The extends the basic STAAD.Pro application with additional capabilities such as an eigen-based advanced buckling analysis, geometric non-linear analysis, and other advanced analysis methods, a faster analysis solver, and access to advanced concrete design using RCDC.

**Primary Licenses**
This displays the status of each license type associated with your account.

**Additional License**
Select additional licenses to use:

- Structural SELECT Entitlements
- STAAD.Beava (Bridge Codes)
- RAM Connection
- Nuclear Design Codes

**Product Activation Wizard**
Opens the CONNECT License Client Activation window, which is used to activate or to reserve a license.

**License Management Tool**
Opens the Bentley Licensing Tool window, which is used to manage your current license entitlements, managed reserved licenses, and review the status of product licenses available to you.

**Help with license selection**
Click to open the latest details on license selection information on Bentley Communities.
Accept
Save the changes to the license configuration and close the dialog.

Exit
Close the dialog without saving any changes to the license configuration.

Related Links
• GS. Starting STAAD.Pro (on page 31)
• GS. Starting STAAD.Pro (on page 31)

GS. Application Configuration dialog
Used to set program-wide parameters for STAAD.Pro.
Opens when Configure is selected on the application Start page.

General tab
Language
Select the user interface language from the drop-down list. You must restart the program to see that language used in user interface.

Base Unit
There are two base unit systems in the program which control the units (length, force, temperature, etc.) in which, values – specifically results and other information presented in the tables and reports – are displayed. The base unit system also dictates what type of default values the program will use when attributes such as Modulus of Elasticity, Density, etc., are assigned based on material types – Steel, Concrete, Aluminum – selected from the program's library. These two unit systems are English (Foot, Pound, etc.) and Metric (KN, Meter, etc.)

Global Axis
Select which axis represents the gravity direction in your model by default. Subsequently all modeling, analysis, and post-processing items would be based on this coordinate system.

Select either the Y up or Z up option.

Note: Some features in STAAD.Pro require that Y up be used for the global axis orientation.

Diagram Background
Select either a White or Black background for the view window. This selection will also set some default colors in which beams, plates and solids are drawn. For example, a white background is accompanied by black lines for drawing beams.

Theme
Select the basic color theme used in the application.

Options tab
Additional Options
Always Save Auxiliary Data
During the course of performing analysis, design, and post-processing operations, multiple files associated with a STAAD project are generated. Set this option to direct the program to save all files in the STAAD project when the project is closed.

Always Delete Results Data If Out of Sync
Set this option to delete results files when a project is opened if the input file has been modified more recently than those input files. This option is helpful in ensuring that results invalidated by recent changes to the input file will not be used in error.
Warn if model accessed from Network Drive
Set this option to display a warning dialog when an input file is opened from a network drive. Using a network drive can dramatically slow performance. It is recommended that local copies of files be made to improve performance.

Remove Bentley (B) logo from Report
Set this option to remove the Bentley logo from reports generated in STAAD.Pro.

**Tip:** You can always add a custom logo and branding to your reports.

**Startup Options**

Show Licensing Information
Check this option during the application launch to be prompted with License Information dialog (see [GS. Starting STAAD.Pro](#) on page 31) listing enabled licensing options.

Show CONNECT Advisor
Check this option to open the CONNECT Advisor window when the program starts.

Show news feed for new posts
Check this option to open the news feed reader when the program starts.

**File Format tab**

**Input format**

Single Line Format
Select whether Joint coordinates as well as Member, Plate, and Solid incidences will be written one per input line or multiple per input line by setting the associated options here.

Write expanded list
Set this option to instruct the program to write out joint, member or element numbers individually, for example: 1 2 3 4 5 instead of 1 TO 5 and consequently, creates voluminous input.

Joint Coordinate Significant Figures
Specify the number of significant digits used for nodal coordinates. This is used for models which require a high degree of precision.

**Output format**

Add key file information
Set this option to instruct the program to list all the key files that were used with the analysis including the date of the principal files.

**Note:** Even if the general case is that the Key Information is displayed, it can be turned off in a particular analysis if the data file contains the command SET NOFILE.

**Error format**

Create Error File
You may choose not to have the error file created at all, though this isn’t advisable. Un-check the box provided for that purpose if you choose to do so.

Log maximum errors per single input
A single line of input may contain more than one error. We can set how many such errors we want shown for each input line.

Log maximum errors
You can also set the maximum number of errors you want reported for the entire file. That number is 100 by default.
Design Codes tab

STAAD.Pro supports several major international design codes. Select the default design code to use when selecting the material-based design in the program.

**Note:** You may not use more than one design code in one single run, even if the Input Command File has more than one `CODE` command. Also note the design code specified by the Design Code tab must match that specified in the Input Command File.

Related Links
- *GS. To change the system units* (on page 40)

GS. Quick Access Toolbar

The quick access toolbar is located just above the ribbon controls. It contains some of the most frequently used tools in a convenient location.

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Save" /></td>
<td>Saves any changes made to the current model. Saves any changes made to the current model.</td>
<td><code>&lt;Ctrl+S&gt;</code></td>
</tr>
<tr>
<td><img src="image" alt="Open" /></td>
<td>Opens the Start page open tab, which is used to select a model to open in the program.</td>
<td><code>&lt;Ctrl+O&gt;</code></td>
</tr>
<tr>
<td><img src="image" alt="Close" /></td>
<td>Closes the current model and returns to the Start page.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Undo" /></td>
<td>Undoes the previous operation.</td>
<td><code>&lt;Ctrl+Z&gt;</code></td>
</tr>
<tr>
<td><img src="image" alt="Redo" /></td>
<td>Undoes the previous undo operation.</td>
<td><code>&lt;Ctrl+Y&gt;</code></td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>---------------------</td>
<td>------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
</tbody>
</table>
| Command File Editor | Opens the current input command file (file extension .std) in the STAAD.Pro Editor. If any change has been made in the structure that has not been saved, you are prompted to save the structure first.  
**Note:** Refer to the [STAAD.Pro Editor help](#) (on page 2251) for additional assistance. |          |
| STAAD Output        | Opens the results of a successful Analysis and Design run in the STAAD.Pro Editor window.                                                                                                                    |          |

**GS. Tool Search**

You can search for any tool by typing part of the name in the Search field.

**Tip:** Hover your mouse pointer over any search result to see the location of the tool in each workflow, ribbon tab, and group.

In the search results drop-down, click **Show Details** to display the tool tip description for each tool in the results list.
GS. Page Control

Within each workflow, there are a series of Pages available which layout the order of that workflow. Though not required, it is recommended to work from left to right along the pages within any workflow.

**Tip:** If you close some of the dialogs for a page and need to display them again, simply click the **Restore View** tool (±) on the right-hand side of the page control bar.

### Analytical Modeling
Pages in the [Analytical Modeling workflow](on page 615):
- Geometry
- Properties
- Materials
- Specifications
- Supports
- Loading
- Analysis
- Design

### Building Planner
Pages in the [Building Planner workflow](on page 895):
- Slab
- Column
- Beam
- Geometry
- Shearwall
- Support
- Release
- Load
- Design

### Piping
Pages in the [Piping workflow](on page 878):
- Pipe Runs
- Supports

### Bridge Deck
Pages in the [Bridge Deck workflow](on page 883):
- Deck
Postprocessing
Pages in the Postprocessing workflow (on page 2213):
- Displacements
- Reactions
- Beam Results
- Plate Results
- Solid Results
- Dynamics
- Reports

Foundation
Pages in the Foundation workflow (on page 2213):
- Foundation

Steel AutoDrafter
Pages in the Steel AutoDrafter workflow (on page 2227):
- Layout
- Drawing
- Material Take Off

Connection Design
Pages in the Connection Design workflow (on page 953):
- Connections
- Results
- Seismic Frame

Advanced Slab Design
Pages in the Advanced Slab Design workflow (on page 1353):
- Envelopes
- Slab Design

Earthquake
Pages in the Earthquake workflow (on page 2240):
- EC8 Stiffness
- EC8 Plans
- EC8 Elevations

Note: The Physical Modeling workflow and Concrete Design workflow do not have pages, but instead launch separate application windows.
**GS. Data Area**

The dialogs and tables to enter and review data for your model open along the right-hand side of the View window.

As you select different workflows and pages in each workflow, the data area updates with the corresponding dialogs and tables for that page.

**GS. View Window**

The graphical view window allows you to view the model or portions of the model using selected views. This is where the model drawings and results are displayed in graphical form. You can use tools to draw your model as well as graphically assign loads or specifications.

**GS. Right-Click Pop-up Menu**

Provides convenient access to frequently used tools and utilities. The menu pops up when the right mouse button is clicked.

*Note: Some of the items in the menu are context-sensitive - that is, they vary depending on previous actions or selected elements in the window.*

**No Selection**

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Used to cut selected object(s) (delete and copy to clipboard). The deleted objects may then be pasted.</td>
</tr>
<tr>
<td>Copy</td>
<td>Used to copy selected object(s) to clipboard for subsequent pasting.</td>
</tr>
<tr>
<td>Paste</td>
<td>Opens the <strong>Paste With Move</strong> dialog, which is used to specify the insertion point for pasting clipboard elements into the model.</td>
</tr>
<tr>
<td>Select Cursor &gt;</td>
<td>Nodes Used to graphically select nodes.</td>
</tr>
<tr>
<td></td>
<td>Beams Used to graphically select beams.</td>
</tr>
<tr>
<td></td>
<td>Plates Used to graphically select plates.</td>
</tr>
<tr>
<td></td>
<td>Surface Used to graphically select surface elements.</td>
</tr>
<tr>
<td></td>
<td>Solids Used to graphically select solids.</td>
</tr>
<tr>
<td></td>
<td>Plates &amp; Solids Used to graphically select plates and solids with one cursor (ignores other object types).</td>
</tr>
</tbody>
</table>
### Menu item | Description
--- | ---
**Geometry** | Used to graphically select nodes, members and elements of the structure simultaneously. To select nodes, members or elements using the Geometry Cursor, simply click on the desired structural components. To select multiple nodes, members and elements, hold `<Ctrl>` while selecting. We may also select the structural components graphically by creating a window on screen with the cursor around these components.

**Physical Member** | Used in Steel Design or Concrete Design to graphically select all those beams defined as a same member in the member set up of member design, simultaneously. Members may be user defined or may be generated automatically using the Auto Form Member tool. To select all the beams defined as a same member using the Member Cursor, just click on one beam. The other beams having the same member name as the selected one will automatically be selected. To select multiple physical members, hold down `<Ctrl>` while selecting. You may also select the physical members graphically by dragging a fence area around these physical members using the cursor.

**Selection Mode >** | Drag Box | Click to activate the drag box selection mode.
Drag Line | Click to activate the drag line selection mode.
Region | Click to activate the region selection mode.

**Take Picture** | Used to take a snapshot image of current view. The picture is automatically added to a picture album.

**Add Beam** | Used to add beam by clicking start and end nodes.

**Tables** | Opens the **Tables** dialog, which is used to display and close different tables, such as Node coordinates, Beam incidences, Node displacements, etc. irrespective of the current page.

**Connection Tags >** | View Connection Tags | Opens the **Assign Connection Tags** dialog.
Check Connection Tags | (Active only after a successful analysis has been performed) Opens the **Assign Connection Tags** dialog and **Check Connection Tags** dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

**Open View** | Opens the **Open View** dialog, which is used to open a previously saved structural view window.
### Menu item | Description
---|---
**Labels**<br>Opens the **Diagrams** dialog to the **Labels** tab, which is used to select various display labels for different components of the structure.<br><br>**Orientation**<br>Opens the **Orientation** dialog, which is used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.<br><br>**Structure Diagrams**<br>Opens the **Diagrams** dialog, which is used to customize the view of the structure by setting different view-related parameters.<br><br>**Model View Details**<br>Opens the **Structural Diagram Info** dialog, which contains counts of the objects in your model.<br><br>**3D Rendering**<br>Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view.<br>Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options.<br>Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.<br><br### Beam Selection

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
</table>
**Move**<br>Opens the **Move Entities** dialog, which is used to specify the translational offset for moving a selection of beams.<br><br>**Insert Node**<br>Opens the **Insert Node into Beam #** dialog, which is used to insert one or more nodes at specified distances along selected members.<br><br>**Form Member**<br>Used to manually form a physical structural member from a selection of one or more connected analytical beam segments.
### Menu item | Description
--- | ---
Properties | Opens the **Beam** dialog which displays properties for the selected member. Additional tabs become available after analysis and design is performed.
Define Section Profile | (disabled)
Connection Tags > Assign | Opens the **Assign Connection Tags** dialog and **New Connection tag** dialog, which are used to assign existing connection tags to member ends in the model and to create new connection tags, respectively.
Remove | Opens the **Assign Connection Tags** dialog and initiates the remove connection tag tool for the connections on the selected member. A confirmation dialog opens to confirm the remove action.
New View | Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way.

**Note:** This option becomes active only after you select one or more structural elements on screen.
Create Seismic Frame | Opens the **Seismic Frames** dialog, which is used to specify the type of seismic frame used for the selected members creating a seismic frame definition during connection design.
Add Member Attribute | Opens the Member Attribute which is used to select and apply a member attribute to selected members.

Refer to [TR.29.1 Struclink Member Attribute](on page 2536) for additional information on the Struclink attribute which is assigned using this dialog.
List/Delete Member Attribute | Opens the Member Attribute dialog to display the currently used Member Attribute for a single selected member.

### GS. Right-Click View Tools menu

Hold **<Shift>** when right-clicking in the View window to display the **View** ribbon tab **Tools** group at the mouse pointer.
### Table 4: Tools group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zoom Window</td>
<td>Used to define the boundaries of a rectangular area of the active view to be displayed within the current view.</td>
<td></td>
</tr>
<tr>
<td>Whole Structure</td>
<td>Fits the current view to display the entire structure limits. Resets view rotation to orthogonal.</td>
<td></td>
</tr>
<tr>
<td>Zoom In</td>
<td>Zoom in the current view by a preset percentage magnification.</td>
<td>scroll wheel up</td>
</tr>
<tr>
<td>Zoom Out</td>
<td>Zoom out the current view by a preset percentage magnification.</td>
<td>scroll wheel down</td>
</tr>
<tr>
<td>Pan</td>
<td>Used to view a different part of the design without changing the view magnification.</td>
<td>Tip: Press and hold the middle mouse button (typically the scroll wheel button) to quickly pan in the view window.</td>
</tr>
<tr>
<td>Zoom Extents</td>
<td>Adjusts the view magnification so that the entire model is visible in the view.</td>
<td></td>
</tr>
<tr>
<td>Zoom Factor</td>
<td>Opens the Zoom Factor dialog, which is used to zoom in or out of the structure by specifying a magnification factor. Factors less than one Zoom out.</td>
<td></td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>---------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td><strong>Dynamic Zoom</strong></td>
<td>Used to define the boundaries of a rectangular area of the active view to be displayed within a new view. The previous view window highlights the location of the zoomed portion.</td>
<td></td>
</tr>
<tr>
<td><strong>Magnifying Glass</strong></td>
<td>Provides a magnified portion of the current view window when you click and drag the pointer.</td>
<td></td>
</tr>
<tr>
<td><strong>Isometric View</strong></td>
<td>(Default view) View the model as an isometric projection.</td>
<td></td>
</tr>
<tr>
<td><strong>Front View</strong></td>
<td>View the structure model from the positive Z axis.</td>
<td></td>
</tr>
<tr>
<td><strong>Left View</strong></td>
<td>View the structure model from the negative X axis.</td>
<td></td>
</tr>
<tr>
<td><strong>Top View</strong></td>
<td>View the foundation mode in plan; from the positive Y axis.</td>
<td></td>
</tr>
<tr>
<td><strong>Back View</strong></td>
<td>View the structure model from the negative Z axis.</td>
<td></td>
</tr>
</tbody>
</table>
### Tool name  |  Description  |  Shortcut
---|---|---
![Right View]  |  View the structure model from the positive X axis.  |  

**Bottom View**  

![Bottom View]  |  View the structure model from the negative Y axis.  |  

**Rotate Up**  

![Rotate Up]  |  Rotate the structure model forward about the X axis.  |  <↑> (Up arrow key)

**Rotate Left**  

![Rotate Left]  |  Rotate the structure model forward about the Y axis.  |  <←> (Left arrow key)

**Spin Left**  

![Spin Left]  |  Rotate the structure model forward about the Z axis.  |  <Ctrl+←> (Ctrl+Left arrow keys)

**Rotate Down**  

![Rotate Down]  |  Rotate the foundation mode backward about the X axis.  |  <↓> (Down arrow key)

**Rotate Right**  

![Rotate Right]  |  Rotate the structure model backward about the Y axis.  |  <→> (Right arrow key)
<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spin Right</td>
<td>Rotate the structure model backward about the Z axis.</td>
<td>&lt;Ctrl+→&gt; (Ctrl+Right arrow keys)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Toggle View Rotation Mode</td>
<td>Used to select an node as the center of rotation. When toggled on, pressing &lt;Ctrl+Shift&gt; and clicking a node will set that node as the center of rotation.</td>
<td></td>
</tr>
<tr>
<td>Orientation</td>
<td>Opens the Orientation dialog, which is used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.</td>
<td>&lt;F4&gt;</td>
</tr>
<tr>
<td>Always Fit in Current Window</td>
<td>Instructs the program on what guidelines to use when drawing a selected set of objects on the screen. It displays the selected portion of the model to a size governed by optimum usage of the dimensions of the current window. When you select various tools the program or switch workflows the size of the drawing window frequently changes. With this tool turned on, the model or selected portions of it, will be drawn in such a manner that all the entities will be drawn within the bounds of the drawing area. This means, the size to which the entities are drawn will correspondingly increase or decrease. With this tool turned off, the size of the entities will remain constant, but that means it may or may not fit within the bounds of the drawing window.</td>
<td></td>
</tr>
</tbody>
</table>

**GS. Quick Commands Pop-up menu**

This menu Opens when you press the space bar when the View window has the application focus.
You can customize the Quick Commands with any tool by clicking the icon and then adding tools from the Customize Quick Commands Popup dialog.

Table 5: Selection group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry Cursor</td>
<td>Used to graphically select nodes, members and elements of the structure simultaneously. To select nodes, members or elements using the Geometry Cursor, simply click on the desired structural components. To select multiple nodes, members and elements, hold &lt;Ctrl&gt; while selecting. We may also select the structural components graphically by creating a window on screen with the cursor around these components.</td>
</tr>
<tr>
<td>All Geometry</td>
<td>Selects all nodes, members, elements, and solids in the model.</td>
</tr>
<tr>
<td>Inverse Geometry Selection</td>
<td>Used to select all objects but the ones which are currently selected.</td>
</tr>
<tr>
<td>Geometry List</td>
<td>Opens the Select Geometry dialog, which is used to select one or more model entities (other than nodes) from a list of all entities in the model.</td>
</tr>
<tr>
<td>Entities Parallel To &gt; XY</td>
<td>Used to select all beams, elements, and surfaces which are parallel to a specified global axis. Select the desired global axis from the sub-menu.</td>
</tr>
<tr>
<td>Geometry Connecting To &gt; Node</td>
<td>Used to find out all the entities that are connected to (i.e., have a common node with) any particular node, beam, plate, or solid. A dialog opens prompting you to select the node, beam, plate, or solid number. You can select to apply immediately or click OK to see all the model entities that are connected to the selection.</td>
</tr>
<tr>
<td>Highlight Entities Sequentially</td>
<td>Opens the Visual Check dialog, which is used to sequentially highlight a specific group of entities (beams, plates, or solids) in numerical order. Controls on the speed of the selection are also included.</td>
</tr>
<tr>
<td>Solids Cursor</td>
<td>Used to graphically select solids.</td>
</tr>
<tr>
<td>All Solids</td>
<td>Selects all solids in the model.</td>
</tr>
<tr>
<td>Inverse Solids Selection</td>
<td>All solids in the selection set are deselected and any previously unselected solids are added to the selection set.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
</tr>
<tr>
<td><strong>Solids List</strong></td>
<td>Opens the <strong>Select Solids</strong> dialog, which is used to select one or more solids from a list of all solids in the model.</td>
</tr>
<tr>
<td><strong>Solids Connecting To &gt;</strong></td>
<td>Select solids that connect to the selected object type.</td>
</tr>
<tr>
<td>Node</td>
<td>Beam</td>
</tr>
<tr>
<td><strong>Members</strong></td>
<td>Used in Steel Design or Concrete Design to graphically select all those beams defined as a same member in the member set up of member design, simultaneously. Members may be user defined or may be generated automatically using the Auto Form Member tool. To select all the beams defined as a same member using the Member Cursor, just click on one beam. The other beams having the same member name as the selected one will automatically be selected. To select multiple physical members, hold down <code>&lt;Ctrl&gt;</code> while selecting. You may also select the physical members graphically by dragging a fence area around these physical members using the cursor.</td>
</tr>
<tr>
<td><strong>Nodes Cursor</strong></td>
<td>Used to graphically select nodes.</td>
</tr>
<tr>
<td><strong>All Nodes</strong></td>
<td>Selects all nodes in the model.</td>
</tr>
<tr>
<td><strong>All Supports</strong></td>
<td>Selects all supported nodes in the model.</td>
</tr>
<tr>
<td><strong>Inverse Node Selection</strong></td>
<td>All nodes in the selection set are deselected and any previously unselected nodes are added to the selection set.</td>
</tr>
<tr>
<td><strong>Nodes List</strong></td>
<td>Opens the <strong>Select Nodes</strong> dialog, which is used to select one or more nodes from a list of all nodes in the model. A free list of node numbers may also be specified.</td>
</tr>
<tr>
<td><strong>Beams Cursor</strong></td>
<td>Used to graphically select beams.</td>
</tr>
<tr>
<td><strong>All Beams</strong></td>
<td>Used to select all beams.</td>
</tr>
<tr>
<td><strong>Inverse Beam Selection</strong></td>
<td>All beams in the selection set are deselected and any previously unselected beams are added to the selection set.</td>
</tr>
<tr>
<td><strong>Beam List</strong></td>
<td>Opens the <strong>Select Beams</strong> dialog, which is used to select one or more beams from a list of all beams in the model.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>**Plates Parallel To &gt; X</td>
<td>Y</td>
</tr>
<tr>
<td>**Beams Connecting To &gt; Node</td>
<td>Beam</td>
</tr>
<tr>
<td><strong>Plates Cursor</strong></td>
<td>Used to graphically select plates.</td>
</tr>
<tr>
<td><strong>All Plates</strong></td>
<td>Selects all plates in the model.</td>
</tr>
<tr>
<td><strong>Inverse Plate Selection</strong></td>
<td>All plates in the selection set are deselected and any previously unselected plates are added to the selection set.</td>
</tr>
<tr>
<td><strong>Plate List</strong></td>
<td>Opens the Select Plates dialog, which is used to select one or more plates from a list of all plates in the model.</td>
</tr>
<tr>
<td>**Plates Parallel To &gt; XY</td>
<td>YZ</td>
</tr>
<tr>
<td>**Plates Connecting To &gt; Node</td>
<td>Beam</td>
</tr>
<tr>
<td><strong>Text Cursor</strong></td>
<td>Used to add comments and titles to pictures and result diagrams. The added text can be plotted, too.</td>
</tr>
<tr>
<td></td>
<td>The inserted text can be deleted, moved and modified using the text cursor. Refer to the Insert Text on the Utilities ribbon for a detailed description on inserting text and modifying it using the text cursor.</td>
</tr>
<tr>
<td><strong>Filtered Selection Cursor</strong></td>
<td>Used to select multiple types of geometric entities (nodes, beams, surfaces, etc.) with specific attributes in one pass. This will reduce the time required to create new views and help quickly identify the location of certain entities on your structure.</td>
</tr>
<tr>
<td><strong>Note:</strong> Before this cursor can be used, the actual filter parameters must be defined in advance in the Selection Filters dialog.</td>
<td></td>
</tr>
<tr>
<td><strong>Group Selection</strong></td>
<td>Opens the Select Groups dialog, which is used to select objects in a named group.</td>
</tr>
<tr>
<td>Tool Name</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>By Property Name</td>
<td>Used to select nodes and members based on specifications associated with them. A number of specification types are included in the sub-menu list.</td>
</tr>
<tr>
<td>Missing Properties</td>
<td>Used to select beams, plates, or solids that lack critical input data (Property, Density, Elasticity, Poisson’s ratio, and Alpha) in the active view.</td>
</tr>
<tr>
<td>Drag Box</td>
<td>Click to activate the drag box selection mode.</td>
</tr>
<tr>
<td>Drag Line</td>
<td>Click to activate the drag line selection mode.</td>
</tr>
<tr>
<td>Region</td>
<td>Click to activate the region selection mode.</td>
</tr>
<tr>
<td>Previous Selection Cursor</td>
<td>Selects the last object(s) selected (if they have been deselected).</td>
</tr>
<tr>
<td>Loads Cursor</td>
<td>Used to modify any load already applied on the model by double clicking it. When selected, the mouse pointer changes to the Load Edit Cursor.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>----------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Labels Settings</td>
<td>Opens the <strong>Diagrams</strong> dialog to the <strong>Labels</strong> tab, which is used to customize the view of the structure by setting different view-related parameters.</td>
</tr>
<tr>
<td>Node Labels</td>
<td>Select any of the label types from the drop-down list to turn the display of that label on or off. Tip: Additional labels are controlled on the Diagrams dialog Labels tab.</td>
</tr>
<tr>
<td>Beam Labels</td>
<td></td>
</tr>
<tr>
<td>Plate Labels</td>
<td></td>
</tr>
<tr>
<td>Solid Labels</td>
<td></td>
</tr>
<tr>
<td>Individual Node Labels</td>
<td></td>
</tr>
<tr>
<td>Individual Beam Labels</td>
<td></td>
</tr>
</tbody>
</table>
### Table 7: Display group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load</td>
<td>Select the active load case, load combination, or load envelope from the pop-up dialog.</td>
</tr>
<tr>
<td>View Loading Diagram</td>
<td>Click to toggle the display of the current load case on the structure.</td>
</tr>
</tbody>
</table>

### Table 8: Geometry Tools group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structure Tools &gt;</td>
<td><strong>Check Multiple Structures</strong>&lt;br&gt;Opens the <strong>List of Structures</strong> dialog, which is used to determine if the current model consists of more than one unconnected structure. &lt;br&gt;Select a Structure in the list to see each structure highlighted.</td>
</tr>
<tr>
<td></td>
<td><strong>Check Beam Plate Connectivity</strong>&lt;br&gt;Used to check for plates that are improperly connected to beams. Beams must be connected to plates at their nodes in order to ensure proper coupling and load transfer. This tool will inform the user if any improper connections are present in the model.</td>
</tr>
<tr>
<td>Tool Name</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Merge Properties</strong></td>
<td>Used to merge the properties of two or more similar objects. When a STAAD input file has a number of references of the same property, this tool can be used to consolidate all these properties into a single command. When a STAAD file has a number of references of the same property, there is now a tool to consolidate all these properties into a single command. Clicking the Yes button, all instances of a given section property will be collated into a single property reference. Properties references with differing additional parameters will not be collated. Properties references with differing assigned material properties will not be collated.</td>
</tr>
<tr>
<td><strong>Cut Section</strong></td>
<td>Opens the Section dialog, which is used to cut a section through the structure along a specified global plane at a desired location of the 3rd axis.</td>
</tr>
<tr>
<td><strong>Check Duplicate Nodes</strong></td>
<td>Opens the Remove Duplicate Nodes dialog, which is used to set the tolerance distance between two nodes the program should consider as duplicate. Used for detecting the presence of two or more instances of the same node.</td>
</tr>
<tr>
<td><strong>Highlight Orphan Nodes</strong></td>
<td>Highlights all nodes in the structure which are not connected to any member, element, or solid.</td>
</tr>
<tr>
<td><strong>Remove Orphan Nodes</strong></td>
<td>Used to remove all detected orphan nodes.</td>
</tr>
<tr>
<td><strong>Node to Node Distance</strong></td>
<td>Display the distance between nodes.</td>
</tr>
<tr>
<td><strong>Remove Node to Node Distance</strong></td>
<td>Used to remove the display of all node to node dimensions from the current view.</td>
</tr>
<tr>
<td><strong>Check Duplicate Beams</strong></td>
<td>Opens the Remove Duplicate Beams dialog, which is used to set the tolerance distance between two beams the program should consider as duplicate. Used for detecting the presence of two or more instances of the same beam.</td>
</tr>
<tr>
<td>Tool Name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Check Zero Length Members</strong></td>
<td>Opens the <strong>Zero Length Tolerance</strong> dialog, which is used to set the tolerance for zero length members. A member connected between duplicate nodes, which have the same (X,Y,Z) coordinates, will have a length of zero. This tool detects such members.</td>
</tr>
<tr>
<td><strong>Check Overlapping Collinear Members</strong></td>
<td>When two members are collinear, and further, at least one of the nodes of one of those members happens to lie within the span of the other, but the two members are not connected at that node, those two members are considered as overlapping collinear members. This tool detects such members. The usefulness of this tool comes from the fact that it enables you to detect modeling errors which are not easily visible. Two lines overlapping on the drawing area become indistinguishable on large models, and this is one of the tools that can spot such errors.</td>
</tr>
<tr>
<td><strong>Redefine Incidence</strong></td>
<td>Used to reverse the incidence of selected beams so that the global coordinates of the end node are farther from the origin than that of the start node.</td>
</tr>
<tr>
<td><strong>Dimension Beams</strong></td>
<td>Display the dimension of the members in the structure.</td>
</tr>
<tr>
<td><strong>Plate Tools &gt;</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Check Duplicate Plates</strong></td>
<td>Opens the <strong>Remove Duplicate Plates</strong> dialog, which is used to set the tolerance distance between two plates the program should consider as duplicate. Used for detecting the presence of two or more instances of the same plate.</td>
</tr>
<tr>
<td><strong>Check for Warped Plates</strong></td>
<td>A warped plate is defined as a four-noded plate whose nodes do not lie on the same plane. This tool detects such plates.</td>
</tr>
<tr>
<td><strong>Check Improperly Connected Plates</strong></td>
<td>Checks the model for plates that overlap or intersect each other at their boundaries and, if found, opens the Overlapping Collinear Plates dialog. Typically, these overlaps cause improper load transfers or instabilities in the model.</td>
</tr>
<tr>
<td><strong>Solid Tools &gt;</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Check for Solids with Negative Volume (Jacobian)</strong></td>
<td>Used to verify that the solid elements in their model have the proper sequence (order) of node numbering to prevent warnings in the output file of solids containing negative volumes.</td>
</tr>
<tr>
<td><strong>Check for Warped Solids</strong></td>
<td>Used to check if a solid element is warped. Warped solids cannot be analyzed and will produce errors in the output file.</td>
</tr>
</tbody>
</table>
### Tool Name | Description
--- | ---

**Physical Member Restraints** | Used to automatically generate top and bottom flange restraint conditions for the selected physical members.

*Note:* The PBRACE command generated by this menu item is valid only for [Member Design](on page 1649) per AS 4100-1998 (Australian) steel design.

When selected, the program will search through all the nodes along the Physical Member and determine if any other beam is connected besides the beams in the current physical member definition. If another beam is present, a full restraint is placed on both the flanges. Otherwise, an unrestrained condition is imposed on both the flanges.

**Groups** | Opens the [Create Group](dialog, which is used to cluster a set of joints, beams, plates or solids into a single entity identified by a distinct name.

*Note:* If no groups exist, you will be prompted to create a group using the [Define Group Name](dialog.

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
</table>
| ![Connection Tags >]( | **Load Connection Tag File**
| ![Edit Connection Tag File]( | **Edit Connection Tag File**
| ![View Connection Tags]( | **View Connection Tags**
| ![Check Connection Tags]( | (Active only after a successful analysis has been performed) Opens the [Assign Connection Tags](dialog and [Check Connection Tags](dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

**Calculator** | Opens the [STAAD.Pro Calculator](window, which is capable of performing mathematical operations.

---
# Getting Started

## GS. Keyboard Shortcuts

Shortcut keys for menu commands are indicated in the menus and/or related dialog boxes.

**Note:** Most ribbon tabs are accessible by pressing the `<Alt>` key and then the letter key corresponding in the tooltip over that ribbon tab. STAAD.Pro also supports shortcut keys for window and dialog box navigation and other functions.

<table>
<thead>
<tr>
<th>Table 10: Working with files</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Action</strong></td>
</tr>
<tr>
<td>Create a new STAAD.Pro project file.</td>
</tr>
<tr>
<td>Open an existing STAAD.Pro project file.</td>
</tr>
</tbody>
</table>
### Table 11: Editing models

<table>
<thead>
<tr>
<th>Action</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Save the current project file.</td>
<td>&lt;Ctrl+S&gt;</td>
</tr>
<tr>
<td>Print the output report.</td>
<td>&lt;Ctrl+P&gt;</td>
</tr>
<tr>
<td>Quit the program.</td>
<td>&lt;Alt+F4&gt;</td>
</tr>
</tbody>
</table>

### Table 12: Navigating the view window

<table>
<thead>
<tr>
<th>Action</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo an action.</td>
<td>&lt;Ctrl+Z&gt;</td>
</tr>
<tr>
<td>Redo an action</td>
<td>&lt;Ctrl+Y&gt;</td>
</tr>
<tr>
<td>Cancel an action.</td>
<td>&lt;Esc&gt;</td>
</tr>
<tr>
<td>Copy a selected element.</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
<tr>
<td>Paste an element from the clipboard.</td>
<td>&lt;Ctrl+V&gt;</td>
</tr>
<tr>
<td>Cut an element (copies the selected element to the clipboard and deletes the selected copy).</td>
<td>&lt;Ctrl+X&gt;</td>
</tr>
<tr>
<td>Delete a selected element.</td>
<td>&lt;Del&gt;</td>
</tr>
<tr>
<td>Move selected objects.</td>
<td>&lt;F2&gt;</td>
</tr>
<tr>
<td>Rotate model up or down in the view window.</td>
<td>Up or Down arrow keys (&lt;↑&gt;, &lt;↓&gt;)</td>
</tr>
<tr>
<td>Rotate model left or right in the view window.</td>
<td>Left or Right arrow keys (&lt;←&gt;, &lt;→&gt;)</td>
</tr>
<tr>
<td>Spin mode left or right in the view window.</td>
<td>&lt;Ctrl+Left&gt; or &lt;Ctrl+Right&gt; arrow keys</td>
</tr>
<tr>
<td>Close the current active view.</td>
<td>&lt;Ctrl+F4&gt;</td>
</tr>
<tr>
<td>Open the Orientation dialog, which is used to precisely control the zoom and rotation of the active view window.</td>
<td>&lt;F4&gt;</td>
</tr>
<tr>
<td>Refresh the active view window.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Tile all view windows, tables, forms, etc. horizontally.</td>
<td>&lt;Shift+F4&gt;</td>
</tr>
<tr>
<td>Cascade all view windows, tables, forms, etc.</td>
<td>&lt;Shift+F5&gt;</td>
</tr>
</tbody>
</table>
Table 13: Working with models

<table>
<thead>
<tr>
<th>Action</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tile all view windows, tables, forms, etc. vertically.</td>
<td>&lt;Ctrl+Shift+F4&gt;</td>
</tr>
<tr>
<td>Animation in full screen (Postprocessing mode).</td>
<td>&lt;F12&gt;</td>
</tr>
</tbody>
</table>

Create a new group from selected entities.  
Initiate the analysis and, if used, batch design for the current input file.  
Open the relevant help topic.

Table 14: Toggling the display of model labels

<table>
<thead>
<tr>
<th>Action</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display member specifications (i.e., truss, beta angle, etc.).</td>
<td>&lt;Shift+A&gt;</td>
</tr>
<tr>
<td>Show axes window.</td>
<td>&lt;Ctrl+Shift+A&gt;</td>
</tr>
<tr>
<td>Display beam numbers.</td>
<td>&lt;Shift+B&gt;</td>
</tr>
<tr>
<td>Display the design brief for physical members.</td>
<td>&lt;Ctrl+Shift+B&gt;</td>
</tr>
<tr>
<td>Display solid element numbers.</td>
<td>&lt;Shift+C&gt;</td>
</tr>
<tr>
<td>Display dimensions.</td>
<td>&lt;Shift+D&gt;</td>
</tr>
<tr>
<td>Display the beam ends, color coded for start and end.</td>
<td>&lt;Shift+E&gt;</td>
</tr>
<tr>
<td>Display the design envelope for physical members.</td>
<td>&lt;Ctrl+Shift+E&gt;</td>
</tr>
<tr>
<td>Display floor loading.</td>
<td>&lt;Shift+F&gt;</td>
</tr>
<tr>
<td>Show the diagram information.</td>
<td>&lt;Shift+G&gt;</td>
</tr>
<tr>
<td>Display the design group for physical members.</td>
<td>&lt;Ctrl+Shift+G&gt;</td>
</tr>
<tr>
<td>Display the wind load tributary area.</td>
<td>&lt;Shift+H&gt;</td>
</tr>
<tr>
<td>Show axes at origin.</td>
<td>&lt;Shift+I&gt;</td>
</tr>
<tr>
<td>Display the entity reference numbers.</td>
<td>&lt;Shift+J&gt;</td>
</tr>
<tr>
<td>Display node point labels.</td>
<td>&lt;Shift+K&gt;</td>
</tr>
<tr>
<td>Action</td>
<td>Shortcut</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>Display the master-slave links.</td>
<td>&lt;Shift+L&gt;</td>
</tr>
<tr>
<td>Display the material label for each entity.</td>
<td>&lt;Shift+M&gt;</td>
</tr>
<tr>
<td>Display physical member numbers.</td>
<td>&lt;Ctrl+Shift+M&gt;</td>
</tr>
<tr>
<td>Display node numbers.</td>
<td>&lt;Shift+N&gt;</td>
</tr>
<tr>
<td>Display the local beam axis (beam orientation).</td>
<td>&lt;Shift+O&gt;</td>
</tr>
<tr>
<td>Display plate element numbers.</td>
<td>&lt;Shift+P&gt;</td>
</tr>
<tr>
<td>Display surface element numbers.</td>
<td>&lt;Ctrl+Shift+P&gt;</td>
</tr>
<tr>
<td>Display member/element releases.</td>
<td>&lt;Shift+R&gt;</td>
</tr>
<tr>
<td>Display the support node labels.</td>
<td>&lt;Shift+S&gt;</td>
</tr>
<tr>
<td>Display the local plate element axis (plate orientation).</td>
<td>&lt;Shift+T&gt;</td>
</tr>
<tr>
<td>Display the local surface element axis (surface orientation).</td>
<td>&lt;Ctrl+Shift+T&gt;</td>
</tr>
<tr>
<td>Display load values.</td>
<td>&lt;Shift+V&gt;</td>
</tr>
<tr>
<td>Display wind loads.</td>
<td>&lt;Shift+W&gt;</td>
</tr>
<tr>
<td>Display member sections.</td>
<td>&lt;Shift+X&gt;</td>
</tr>
<tr>
<td>Display the floor load distribution.</td>
<td>&lt;Shift+Y&gt;</td>
</tr>
</tbody>
</table>

**Table 15: Model rendering**

<table>
<thead>
<tr>
<th>Action</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>View the analytical model (no outline)</td>
<td>&lt;Ctrl+0&gt;</td>
</tr>
<tr>
<td>View the model in wireframe (outline of members)</td>
<td>&lt;Ctrl+1&gt;</td>
</tr>
<tr>
<td>View the model filled (members drawn solid)</td>
<td>&lt;Ctrl+2&gt;</td>
</tr>
<tr>
<td>View the rendered model window</td>
<td>&lt;Ctrl+4&gt;</td>
</tr>
</tbody>
</table>

**GS. Services and Support Information**

These resources are provided to help you get answers for your technical questions on STAAD.Pro.
Service Request Manager

http://apps.bentley.com/srmanager/ProductSupport — Create and track a service request using Bentley Systems’ online site for reporting problems or suggesting new features.

Communities Support

https://communities.bentley.com/support — Find solutions to issues, get answers to questions, or get priority assistance for critical issues.

FAQ and TechNotes

https://communities.bentley.com/products/structural/structural_analysis___design/w/structural_analysis_and_design_wiki/4057.staad-pro-support-solutions

Forums

http://communities.bentley.com/products/structural/structural_analysis___design/f/5932 — Post questions in the Bentley Communities forums to receive help and advice from fellow users and members of Bentley’s product support groups.

Nuclear Safety Related Features

An installation of STAAD.Pro provides users with an extensive set of features and capabilities. It is recognized that STAAD.Pro can be used on projects that require the highest level of safety such as nuclear power stations. Therefore, a set of key features as listed below have been defined as “Nuclear Safety Related Features.” For any user who has signed up to the QA&R program, any serious defect discovered in any of these features in a build of STAAD.Pro supplied as part of the QA&R that is determined to be of Critical or High Severity, will be notified as per the reporting requirements.

Models must be defined using the default axis system (Y-up) and the following capabilities are identified as being safety related.

1. Static Analysis
   a. first order elastic analysis
   b. P-Delta analysis
2. Dynamic Analysis
3. Features related to application of general loadings applied at/on nodes, members, plates and solids.
4. Features related to application of local boundary conditions (e.g., member releases, offsets etc.)
5. Features related to application of global boundary conditions (e.g., fixed, pinned, fixed but, spring supports)
6. Automated load generation features as per US codes (e.g., ASCE, IBC, UBC) such as:
   a. Wind load generation per ASCE 7
   b. Seismic load generation
   c. Response spectrum load generation
   d. Moving load generation

Design Codes:
1. The following STAAD command file driven Steel design are available:
   a. AISC 360-2016
   b. AISC 360-2010
   c. AISC 360-2005
   f. AISC N690 1984 and 1994 with Supplements S1 and S2
   h. CAN/CSA S16-14
   i. CAN/CSA S16-09

2. The following STAAD command file driven Concrete design are available:

Note that any other features, such as the graphical user interface and interfaces to link to external applications such as AutoPipe, RAM Connection, RAM Concept etc. are not considered as “Nuclear Safety Related Features” of the product/program.
The Software Release Report for STAAD.Pro contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 06. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

STAAD.Pro CONNECT Edition V22

CONNECT Edition V22 Update 1

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 1 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 (release 22.00). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.01.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.01.00-1.1 Reduced Section Properties per IS1893 2016

IS1893 2016 Clause 6.4.3 calls for a reduction of moment of inertia values by 0.35 for beams and 0.7 for columns of concrete members only while analyzing the structure for static seismic and response spectrum/linear dynamic analysis. STAAD.Pro now includes a code-specific option for reduced section properties of concrete members which will automatically generate a separate stiffness matrix for these IS1893 2016 load cases. For all other load cases, the analysis will be performed using the unreduced stiffness matrix.

You may use this feature by either selecting the new Code Specific option and then selecting IS1893 2016 as the code in the Member Specification dialog Property Reduction Factors tab. You can manually enter the command using the MEMBER CRACKED CODE IS1893 2016 command.

Note: Automated stiffness reduction analysis is not supported by the basic solver. STAAD.Pro Advanced is required for this feature.

Using the code-specific member reduction properties helps by automating the analysis for when reduced section properties are required by the IS1893 2016 code. A separate stiffness matrix is produced using the reduced stiffness values for use with load cases containing IS1893 2016 static seismic and response spectrum loads. The unreduced stiffness matrix is then used for all other load cases.
The prime advantage of this feature is that this analysis is completely automated once the reduction factors are defined. Previously, in order to comply with the IS1893 2016 Clause 6.4.3 provision, you would have to specify reduced section properties for members, analyze the structure for only the IS1893 2016 seismic load cases, then use a CHANGE command to reset the stiffness reduction factors back to unity, and then re-analyze the structure for the remaining load cases. Care had to be taken that only concrete members had the reduction factor applied, as the global MEMBER CRACKED command can be applied to any member. For large models with hundreds and thousands of members and hundreds of load cases, the implementation of clause 6.4.3 using the previous reduction command would be very cumbersome.

Use of the code-specific reduction factors automates the analysis procedure. These reduction factors are only applied to concrete members with the member list or member groups (non-concrete members are ignored for this command). The separate stiffness matrix is automatically created using these reduction factors for use with IS1893 2016 seismic loads only while the unreduced stiffness matrix is used for all other load cases. No re-analysis or change commands are required.

Related Links
- Member Specification dialog (on page 2962)
- TR.20.10 Member Property Reduction Factors (on page 2485)
- M. To assign cracked section properties to a member (on page 802)

RR 22.01.00-1.2 Import General Sections from Section Wizard

General shapes can now be imported from Section Wizard for use in the program.

Note: This profile shape should consist of only an outer contour. Any internal voids such as internal contours or openings will be ignored, though section properties calculated by Section Wizard based on sections with voids will be used.

Tip: Changing the structure diagram to display 3D rendering types for Full Sections or Sections Outline will display the shape profile. Similarly, 3D renderings of the structure will also display the shape profile.
RR 22.01.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.01.00-2.1 Physical Modeler Static Seismic Loads

STAAD.Pro Physical Modeler can now generate static (equivalent lateral force) seismic load cases for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using static seismic loads.

RR 22.01.00-2.2 Physical Modeler Response Spectra Loads

STAAD.Pro Physical Modeler can now generate response spectra load cases for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using response spectrum loads.
RR 22.01.00-2.3 Physical Modeler Time History Loads

STAAD.Pro Physical Modeler can now generate time history load cases for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using time history loads.

RR 22.01.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.01.00-3.1 Static Seismic Loads per IBC 2018 / ASCE 7-16

Automatically generated equivalent lateral seismic loads per IBC 2018 / ASCE 7-16 are now available in STAAD.Pro.

The load parameters are assigned using the User Interface or can be manually entered in the STAAD editor.

Automatic load combinations using the tables from IBC 2018 can be added using the Auto Load Combination dialog (on page 3033), as well.

Changes from IBC 2015/ASCE 7-10 Code

The changes in seismic provisions of IBC 2018/ASCE 7-16 over the previous version IBC 2015/ASCE 7-10 are mainly observed in Mapped Spectral Acceleration parameters outlined in chapter 22. STAAD.Pro uses the spectral acceleration data in text format available in USGS site https://earthquake.usgs.gov/hazards/designmaps/datasets/. As the IBC 2018/ASCE 7-16 version of the data is not available, IBC 2012/ASCE 7-10 data in text format is used in the IBC 2018/ASCE 7-16 implementation.

The other significant change in the seismic provisions are in long-period site coefficient, $F_a$, and short-period site coefficient, $F_v$. Table 11.4-1 and 11.4-2 of the code provides the value of $F_a$ and $F_v$ respectively. These values are used for the IBC 2018 implementation in STAAD.Pro.

Note: Other changes to this portion of the code from the previous edition are not relevant to the calculations performed by STAAD.Pro.

Related Links
- TR.31.2.15 IBC 2018 Seismic Load Definition (on page 2603)

RR 22.01.00-3.2 IS 801 Cold-Formed Steel Design

The design of cold-formed steel members per the IS 801 code is again available in STAAD.Pro.

RR 22.01.00-3.3 IS 1893 2016 Seismic Irregularities Checks

STAAD.Pro can now check for vertical and plan irregularities per the IS 1893 2016 code for structures with defined rigid floor diaphragms.

Note: The program can additionally check for soft stories per the IS 1893 2016 code.

Horizontal irregularities (torsional and reentrant corners) per Table 5 and vertical irregularities (mass irregularities and irregular modes of oscillation) per Table 6 of IS 1893 2016 can be checked by the program using this command.
**Example Output**

**-IRREGULARITY CHECKS**

STAAD.PRO IRREGULARITIES CHECK - ( IS1893-2016 ) v1.0

******************************************************************************

---TORSION IRREGULARITY CHECKS

Torsion Irregularity Check
Ref: Table 5 (i) - Ratio Limit: 1.50

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Extreme Points of Dia in X</th>
<th>Extreme Points of Dia in Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>(mm)</td>
<td>(mm)</td>
<td>(mm)</td>
</tr>
<tr>
<td>1</td>
<td>12</td>
<td>6</td>
</tr>
<tr>
<td>2</td>
<td>36</td>
<td>30</td>
</tr>
</tbody>
</table>

Diaphragm DX-max/min DZ-max/min Status

<table>
<thead>
<tr>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>OK</td>
</tr>
<tr>
<td>OK</td>
</tr>
</tbody>
</table>

---GEOMETRY IRREGULARITY CHECKS

Re-Entrant Corner Check.
(Ref: Table 5 (ii) - Ratio Limit: 0.15)

<table>
<thead>
<tr>
<th>Node Connectivity</th>
<th>X-Proj (m)</th>
<th>X-Proj/Lx</th>
<th>Z-Proj (m)</th>
<th>Z-Proj/Lz</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-&gt; 1</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>5</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td></td>
</tr>
<tr>
<td>6-&gt; 7</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>8</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>9-&gt; 10</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td></td>
</tr>
<tr>
<td>12-&gt; 4</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>3</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>26-&gt; 25</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>29</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td></td>
</tr>
<tr>
<td>30-&gt; 31</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>32</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>33-&gt; 34</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>35</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td></td>
</tr>
<tr>
<td>36-&gt; 28</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>27</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
</tbody>
</table>

Diaphragm: Lx: (m)  Lz: (m)

<table>
<thead>
<tr>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

---MASS IRREGULARITY CHECKS

Mass Irregularity Check
Ref: Table 6 (ii) - Ratio Limit: 1.50

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Level</th>
<th>Mass</th>
<th>Above</th>
<th>Below</th>
<th>Ratio</th>
<th>Ratio</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>(m)</td>
<td>(kN)</td>
<td>(kN)</td>
<td>(kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.000</td>
<td>636.163</td>
<td>459.451</td>
<td>Base</td>
<td>1.385</td>
<td>N/A</td>
<td>OK</td>
</tr>
<tr>
<td>2</td>
<td>5.000</td>
<td>459.451</td>
<td>Top</td>
<td>636.163</td>
<td>N/A</td>
<td>0.722</td>
<td>OK</td>
</tr>
</tbody>
</table>

***NOTE: Linear dynamic analysis needs to be carried out for Irregular Modes of Oscillation check.

***NOTE: Static Seismic Loads for relevant code needs to be defined with Zone 4 and 5 for Irregular Modes of Oscillation check.

Related Links

- TR.28.2 Floor Diaphragm (on page 2526)
- TR.28.2.1 Soft Story Checking (on page 2529)
- Floor Diaphragm Options dialog (on page 3079)
- TR.28.2.2 Check Irregularities (on page 2531)
- A. To check for soft stories and seismic code irregularities (on page 937)

RR 22.01.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.01.00-4.1 Steel AutoDrafter Workflow

The Steel AutoDrafter can now open any drawing directly within MicroStation.

Note: If you have MicroStation PowerDraft installed instead of MicroStation, you will have that product’s icon instead of the MicroStation icon in the drawing toolbar.

Related Links

- To open a drawing in MicroStation (on page 2235)

RR 22.04.00-4.2 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.0 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.0 (CONNECT Edition V13).
- Connection Design per the AS4100 1998 (Australian) code
CONNECT Edition V22

The Software Release Report for STAAD.Pro CONNECT Edition V22 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition Update 3 (release 21.03). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.00.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

**Related Links**
- [GS. STAAD.Pro License Configuration dialog](on page 53)
- [GS. Starting STAAD.Pro](on page 31)

**RR 22.00.00-1.1 CONNECT Licensing**

STAAD.Pro now uses Bentley’s CONNECT License system, Bentley’s new process for product activation and usage tracking, improving our licensing capabilities with features such as:

- License alert notifications when you are approaching a custom usage threshold
- Replacing site activation keys with user validation, enhancing security around your Bentley licenses and subscriptions

With traditional SELECT Licensing, product activation has been through an activation key that an Organization distributed to all users. With CONNECT Licensing, product activation is managed by user sign in through the CONNECTION Client, which is installed on each machine that uses Bentley applications. This offers a more secure and manageable system as it offers usage alerts, notifying your users when they are about to reach a certain usage limit set by the Administrator.

Detailed information is available at Bentley Communities Licensing and Activation Wiki: [https://communities.bentley.com/products/licensing/w/licensing_wiki/37813/connect-licensing](https://communities.bentley.com/products/licensing/w/licensing_wiki/37813/connect-licensing) for detailed information.

**Related Links**
- [GS. STAAD.Pro License Configuration dialog](on page 53)
- [GS. Starting STAAD.Pro](on page 31)

**RR 22.00.00-1.2 Structural Entitlements**

The Steel AutoDrafter and Building Planner Workflows added in STAAD.Pro CONNECT Edition v21.03.00.146 as Technical Preview features are now fully commercial and are licensed with a new ‘Structural SELECT Entitlement’ license.

This license is provided free with a current STAAD.Pro (including STAAD.Pro Advanced) or Structural Enterprise License.

**Related Links**
- [GS. STAAD.Pro License Configuration dialog](on page 53)
STAAD.Pro CONNECT Edition

CONNECT Edition Update 3

The Software Release Report for STAAD.Pro CONNECT Edition Update 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition Update 2 (release 21.00.02). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 21.03.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 21.03.00-1.1 APL Apollo Structural Tube Sections

The catalog of available steel sections has been expanded to include the hollow sections from APL Apollo (India).

To specify an APL Apollo tube section

1. Select the Specifications ribbon tab.
2. On the Properties - Whole Structure dialog, click Section Database.
   
   The Section Profiles Table dialog opens.
3. Select APL Apollo Tubes in the list of tables on the Steel tab.
4. Select the profile type to use from the list of tables:
   
   Rectangular Hollow
   Square Hollow
   Circular Hollow
5. Select the profile from the Select Profile list.
6. (Optional) Check the Material option to specify the selected material with the profile.
7. Click Add.

The section is now added to the Section tab in the Properties - Whole Structure dialog box and can be assigned to members.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
- TR.20 Member Property Specification (on page 2459)
- Section Profile Tables dialog (on page 2969)
- RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10 (on page 94)
RR 21.03.00-1.2 Steel Grade A1085 Profiles in AISC 360-16 and 360-10

Steel pipes and tubes using A1085 grade material can now be designed using AISC 360-16 and AISC 360-10 design codes.

A new set of section tables are available in the American Steel Tables for HSS Rectangle and HSS Round shapes to make use of the A1085 steel grade. Sections using this steel grade have properties which use the full nominal wall thickness.

**Note:** Use of profiles from the A1085 steel tables will result in that material be used, regardless of the SGR parameter for AISC 360-16 or WTYP parameter for AISC 360-10.

**Related Links**
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
- D1.A.4.1 AISC Steel Table (on page 1368)
- D1.A.6 Design Parameters (on page 1378)
- RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10 (on page 94)

RR 21.03.00-1.3 Inclined Loads Input via Create New Load Items Dialog

Inclined loads can now be easily added via the graphical user interface in the Create New Load Items dialog.

Previously, an inclined nodal load –that is a nodal load applied in coordinate system other than the global coordinate system– required editing the STAAD input file. These loads can now be applied by defining a reference node, relative reference point, or absolute reference point in the graphical user interface's Analytical Modeling workflow.

**Related Links**
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
- Nodal Load tab (on page 3008)
- M. To add a nodal load (on page 824)
- RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10 (on page 94)

RR 21.03.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 21.03.00-2.1 Physical Modeling Load Cases and Combinations

STAAD.Pro Physical Modeler can now generate load cases and combinations for use directly in the Analytical Modeling workflow.

RR 21.03.00-2.1 Physical Modeling Circular Openings

STAAD.Pro Physical Modeler can now model circular openings in surfaces with a defined radius.

RR 21.03.00-2.3 Physical Modeling Connection Tags

You can now assign connection tags from with STAAD.Pro Physical Modeler. These will be exported to the analytical model in STAAD.Pro as well as to a corresponding ISM repository.
RR 21.03.00-2.4 Miscellaneous Enhancements to the Physical Modeler

The following additional enhancements have been made to the STAAD.Pro Physical Modeler.

- A new file format is used. When opening a model created in a previous version of STAAD.Pro Physical Modeler, you will be prompted to save to the new format.
- New **Spreadsheet** and **Data** ribbon tabs have been added to make the spreadsheet controls consistent with other tools in the application. The **Spreadsheet** ribbon tab is used to select the current spreadsheet. The **Data** ribbon tab is used to perform actions on the current spreadsheet, view window, or output panel (similar to right-click pop-up menus in each of those respective application areas).
- The ISM workflow has been enhanced to place common settings in the program configuration. This helps to streamline ISM repository actions.

RR 21.03.00-2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

Automatically generated equivalent lateral seismic loads per IBC 2015 / ASCE 7-10 are now available in STAAD.Pro.

The load parameters are assigned using the User Interface or can be manually entered in the STAAD editor. Automatic load combinations using the tables from IBC 2015 can be added using the **Auto Load Combination dialog** (on page 3033), as well.

**Related Links**

**RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10**
- **D1.A.10.5 Combined Forces and Torsion** (on page 1394)
- **P. Steel AutoDrafter Workflow** (on page 2227)
- **D8.F. Indian Codes - Concrete Design per IS 13920-2016** (on page 1955)

**RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10**
- **TR.31.12.14 IBC 2015 Seismic Load Definition** (on page 2600)
- **RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10** (on page 94)

**RR 21.03.00-3.1 Response Spectra per IBC 2015 / ASCE 7-10**

Automatically generated response spectra loads per IBC 2015 / ASCE 7-10 are now available in STAAD.Pro.

**Related Links**

**RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10**
- **TR.32.10.1.10 Response Spectrum Specification per IBC 2015** (on page 2754)
- **RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10** (on page 94)

**RR 21.03.00-3.3 Static Seismic Loads per IS 1893 2016**

Automatically generated equivalent lateral seismic loads per IS 1893 2016 are now available in STAAD.Pro.

**Related Links**

**RR 21.03.00-3.3 Static Seismic Loads per IS 1893 2016**
- **TR.31.2.10 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load** (on page 2582)
RR 21.03.00-3.3 Response Spectra Loads per IS 1893 2016

Automatically generated response spectra loads per IS 1893 2016 are now available in STAAD.Pro.

Related Links
• TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2016 (on page 2732)
• RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10 (on page 94)

RR 21.03.00-3.5 Soft Story Checks per IS 1893 2016

The check soft story option may be added to a floor diaphragm command set to check for soft stories per IS1893:2016. The previous (approximate) method for checking soft stories implemented for IS 1893 2002 has been deprecated in favor of a displacement-based approach using rigid diaphragms.

Soft story checking a process by which designers check stiffness of a story with that of the story above. If the stiffness of a story is lower than that of the story above, the story is considered as a Soft story. Having a soft story in the building makes that story venerable to Earthquake. As per the revised IS 1893 Part-1:2016, if a soft story is detected, structural configuration needs to be change with the introduction of additional lateral load resisting elements in the story detected as soft story. The previous version of IS 1893 Part-1, had provisions to increase design story shear in the soft story and increase percent of steel in the lateral load resisting elements in that soft story. This provision has been withheld in the present version of the code. This makes the situation a stringent one for the designer if a story is detected as a soft story.

Soft Story Checking

There was an approximate methodology available in STAAD.Pro. The program used to compute story stiffness of a story by summing up the lateral stiffness of columns and shear walls (modelled using surface elements). The stiffness of a column is calculated as $12EI / L^2$ where E is the Young's modulus, I is the moment of inertia and L is the length of the column respectively and that for a shear wall (without opening) is calculated as $P_3/3EI + 1.2P_3/h$ (i.e. summation of flexural stiffness and shear stiffness, obtained as deflection of a cantilever wall under a single lateral load P at its top) where h is the height, A is the cross-sectional area and G is the shear modulus of the wall (E and I carry usual meaning).

This method is approximate one and works in the presence of FLOOR HEIGHT commands. On studying the literatures available on this subject, this method proved to be approximate as it fails to capture actual story stiffness including the effect of bracings. Story stiffness should be computed using a displacement based approach. STAAD.Pro is currently equipped with the facility to consider the in-plane stiffness of slabs as Rigid Diaphragm. In STAAD.Pro lateral stiffness is calculated only when the floor is modeled as rigid floor diaphragm since it functions as transferring story shears and torsional moments to lateral force-resisting members during earthquake.

**Story Stiffness Computation Example**

Consider a multi-story building:
The points are master nodes. The story stiffness is defined as the inverse of inter-story drift when a unit load is applied at that story only.

The unit load applied is along X, Z and θY.

Consider the displacement of 2nd story. The displacements are:

Story 1: ΔX1, ΔZ1, θY1;
Story 2: ΔX2, ΔZ2, θY2

Relative Displacement for story 2 is:
RX2 = ΔX2 - ΔX1; RZ2 = ΔZ2 – ΔZ1; RθY2 = θY2 - θY1

Story Stiffness X = 1/((ΔX2 – ΔX1))
Story Stiffness Z = 1/((ΔZ2 – ΔZ1));
Story Stiffness θY = 1/((θY2 - θY1))

The rest is the same for all the other stories.

**Note:** This story stiffness is not the same as column story stiffness which is calculated as \((nCol \times 12EI/L3)\), \(n=\) number of columns.

**Command Input**

The story stiffness can be printed using the PRINT STORY STIFFNESS command.

If soft story check is required to be performed following commands are required to be defined immediately after rigid floor diaphragm is specified in the FLOOR DIAPHRAGM command.

**Related Links**
What's New?

STAAD.Pro CONNECT Edition

**RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10**
- *Floor Diaphragm Options dialog* (on page 3079)
- *TR.28.2.1 Soft Story Checking* (on page 2529)
- *TR.28.2 Floor Diaphragm* (on page 2526)
- *A. To check for soft stories and seismic code irregularities* (on page 937)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 94)

**RR 21.03.00-3.6 Design of I-Sections with Cover Plates for Torsion per AISC 360-16**

STAAD.Pro can now design I-section shapes with top, bottom, or both cover plates for torsion per the AISC 360-16 code.

**Related Links**
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 94)

**RR 21.03.00-3.7 Concrete Design in Metric per ACI 318-14**

STAAD.Pro will now report the concrete design results per ACI 318-14 in metric values when the input is given in metric.

**Related Links**
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 94)

**RR 21.03.00-3.8 Concrete Design per IS13920-2016**

STAAD.Pro is now capable of performing design of concrete beams and columns per IS13920-2016.

**Related Links**
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 94)

**RR 21.03.00-4 Features Affecting Post Processing**

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

**RR 21.03.00-4.1 Steel AutoDrafter Workflow**

The Steel AutoDrafter workflow is used to automatically generate construction drawings and material take-off tables for your steel framed structure.

**Technical Preview:** This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

**Related Links**
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 94)
RR 21.03.00-4.2 Horizontal Brace Connection Design in Connection Design

Connection Design is now capable of designing horizontal bracing connections to beam-beam-brace connections and column-beam-brace connections.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
- D. To design an HBBB connection (on page 969)
- D. To design an HCBB connection (on page 969)
- RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10 (on page 94)

CONNECT Edition Update 2

The Software Release Report for STAAD.Pro CONNECT Edition Update 2 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition Update 1 (release 21.00.01). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 21.00.02-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.
RR 21.00.02-1.1 Select ribbon tab

A new ribbon tab containing tools for selecting model objects has been added to several of the workflows. Having quick access to the selection tools on a single ribbon tab makes selecting objects and then further refining the selection set easier.

The Select ribbon tab is available in multiple workflows, so you no longer have to switch to the Analytical modeling workflow to change the object selection as required in some instances in previous releases.

Related Links

- Select tab (on page 2945)

RR 21.00.02-1.2 New Output Viewer

The STAAD.Pro Editor is now used to view the analysis results file in the Analytical modeling workflow. The updated editor is now used to navigate the results file (file extension .anl).

Related Links

- P. To view analysis results (on page 2212)

RR 21.00.02-2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 21.00.02-2.1 AISC 360-16

The program can now perform steel design of members per ANSI/AISC 360-16 Specifications for Structural Steel Buildings.

The seismic provisions of AISC 341-16 are also implemented.

Performance Enhancements - Design for CODE CHECK / MEMBER SELECT up to 20 times faster than AISC 360-10.

Detailed Design output - TRACK 0,1, & 2 reports improved to display more intermediate results.

AISC 360-16 Design

To use the 2016 edition, specify the command:

CODE AISC UNIFIED 2016

The checking or selection of members is performed per the following sections of AISC 360-16 and AISC 341-16:

- Section Classification: Chapter B
- Tension Chapter D
- Compression: Chapter E
- Flexure: Chapter F
- Shear: Chapter G
- Torsion (Including checks as per DG-9 for certain profile shapes – see below): Chapter H & Design Guide 9 for Torsion design
- Combined actions – Chapter H for:
  - Axial and/or
- Flexure and/or
- Shear and/or
- Torsion
- Checks for Seismic suitability – AISC 341-16

The following section profile shapes are allowed for design:

- Rolled Shapes
  - I-Shape
  - Channel
  - Single angle
  - Solid Circular bar
  - Solid Rectangular bar
  - Tee
  - HSS rectangular
  - HSS round
- Built Up Shapes
  - Built Up Channel
  - Built Up I
  - Double Channels – back to back
  - Double Channels – face to face
  - Double Angles
  - Built Up Box
  - I With Cover Plates (Top and/or bottom)
- Tapered Shapes
  - I with web tapered
  - Square hollow shape with wall tapered
  - Circular hollow shape diameter tapered

**Note:** Certain checks (e.g., DG-9 torsion checks) will only be performed for certain profile shapes (non-HSS shapes).

**RR 21.00.02-2.2 ACI 318-14**

The program can now design concrete columns and beams per ACI 318-14 *Building Code Requirements for Structural Concrete*. The following describes the code clauses from ACI 318-14 implemented for beams and columns.

**Note:** Plate elements and deep beams cannot be designed per ACI 318-14. Use an older edition of the ACI code to design these elements if needed.

**Materials**

- Beams
  
  19.2.2
  19.2.3.1
19.2.4.2

- Columns
  19.2.2
  19.2.4.2

**Minimum Reinforcement**

- Beams
  9.6.1.1
  9.6.1.2
  9.6.1.3
  9.7.2.1
  9.7.6.4.2
  9.7.6.4.3

- Columns
  10.6.1
  10.7.2.1
  10.7.3.1
  10.7.6.1
  10.7.6.2
  25.2.3

Clauses 9.7.6.4.2 and 9.7.6.4.3 are not included in the standard design criteria of the minimum reinforcement since the design of these criteria should consider the rebar including the bar detailing and thus they should be applied in a second design after performing the strength design and the bar detailing.

Clause 9.6.1.3 is also not included in the standard criteria since the reduction of the minimum reinforcement is optional.

**Service**

- Beams
  9.7.2.2
  9.7.2.3

**Strength**

- Beams
  9.5
  9.5.1
  9.5.2
  9.5.3
  9.5.4
  9.6.3.1
  9.6.3.3
  9.6.4.2
What's New?
STAAD.Pro CONNECT Edition

9.7.5.2
9.7.6.2.2
9.7.6.3.1
9.7.6.3.3
9.7.3.3
9.6.4.3
20.2.2.4a
20.2.2.1
20.2.2.2
21.2.1
22.2.2.2
22.5.5
22.2.2.3
22.7.4
22.5.1.2
22.7.6.1
22.7.7.1

- Columns

10.5
10.5.1
10.5.2
10.5.3
10.6.2
10.7.6.5.2
20.2.2.4a
20.2.2.1
20.2.2.2
22.5
22.2.2.2
22.5.5
22.2.2.3
22.7.4
22.5.1.2
22.7.6.1

**Ductility**

- Beams

9.3.3.1

**Development Length**

- Beams

25.4.1.4
25.4.2
RR 21.00.02-2.3 Russian Wind Load per SP 20.13330.2016

Static and dynamic wind loads per the SP 20.13330.2016 SNiP 2.01.07-85 code can now be applied in STAAD.Pro.

In order to perform dynamic wind load generation in STAAD.Pro you must previously define a static load case. The static load case could be any primary static load case which is defined before the Russian dynamic load case, including a static wind loading per the same code. This static load case will provide static load vector to the dynamic wind load module.

As the Russian dynamic wind load component requires modal masses and eigen vectors to calculate the dynamic wind load component at nodes, modal analysis must be performed before the dynamic wind load definition. Therefore, you must also include a separate load case for modal analysis with reference mass defined before the load case. See G.17.3.2 Mass Modeling (on page 2365) for details. Alternatively, if the mass loads are not needed for use with other load cases (that is, a reference load case is not needed for other loads), then you may define the mass loads within the dynamic load case prior to the WIND LOAD command.

Related Links

- TR.31.3 Definition of Wind Load (on page 2623)
- TR.32.12.3 Generation of Wind Loads (on page 2779)
- Wind Load tab (on page 3018)
- M. To add a SNiP wind load definition (on page 837)
- M. To apply a dynamic wind load per SP 20.13330.2016 (on page 839)

CONNECT Edition Update 1

The Software Release Report for STAAD.Pro CONNECT Edition Update 1 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition (release 21.00.00). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.
RR 21.00.01-1.1 Share to ProjectWise

You can now share a STAAD.Pro project directly to a ProjectWise data source. STAAD.Pro lets you select which files associated with the input (.std) file you want to include.

Related Links
- *I. To share a STAAD.Pro project in ProjectWise* (on page 2285)

RR 21.00.01-1.2 Quick Commands

Two new sets of quick commands have been added to the graphical view window to allow for quicker model navigation and modeling.

**Quick Navigation**

Press `<Shift>` when right-clicking in the view window to open the View ribbon tab Tools group and your mouse pointer location.

**Quick Commands**

Press the space bar when the view window has the application focus to open the Quick Commands menu. This customizable pop-up menu contains the following groups of tools:

- **Geometry** ribbon tab Selection group
- **View** ribbon tab Labels group
- **Loading** ribbon tab Display group
- **Utilities** ribbon tab Geometry Tools group
- **Utilities** ribbon tab Utilities group

You can customize the Quick Commands with any tool by clicking the icon and then adding tools from the Customize Quick Commands Popup dialog.
RR 21.00.01-2 Features Affecting the Preprocessor

This section describes features that have been added that affect the preprocessor section of the program, also known as the Modeling Mode.

RR 21.00.01-2.1 Ritz Vector Analysis with User-Defined Starting Load Vectors

You can now directly specify the starting load vectors for use with load dependant Ritz vector method for the eigen solution.

Related Links

- TR.31.9 Defining Starting Load (on page 2648)
- M. To use starting vectors with load-dependant Ritz vectors (on page 865)

RR 21.00.01-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 21.00.01-3.1 AISC 360-16 Technical Preview

The program can now perform steel design of members per ANSI/AISC 360-16 Specifications for Structural Steel Buildings.

Note: This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

Related Links

- D1.A. American Codes - Steel Design per AISC 360 Unified Specification (on page 1366)

CONNECT Edition

The Software Release Report for STAAD.Pro CONNECT Edition contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 6) (release 20.07.11). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 21.00.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 21.00.00-1.1 New User Interface

The STAAD.Pro user interface has been updated and refined.
**Ribbon Controls**

The menus and toolbars have been replaced with a series of tabbed toolbars along the top of the application window. These are context sensitive to the current "workflow" as well as to the current model selection. There are fewer tools displayed at one time, so they can be larger and thus easier to identify. However, the tools you need are always available by selecting the tab (similar to menus in a tradition Windows program).

The icons for the tools are updated to reflect a consistent theme, further making the correct tool easy to identify.

**Workflows Panel**

The system of application modes has been replaced with the Workflows panel.

**Tip:** Once you become familiar with the workflows, you can hide the panel to display a larger viewing area. Simply click the Workflow tab again to select a different workflow.

**Tip:** If you close some of the dialogs for a page and need to display them again, simply click the **Restore View** tool ( ) on the right-hand side of the page control bar.

**Page Control Bar**

The pages within each workflow have been streamlined from those in the previous modes. A single row of pages is displayed for the current workflow, just above the view window. Select the appropriate page for your current task to display the corresponding dialogs and tables.

**RR 21.00.00-1.2 CONNECT Advisor**

STAAD.Pro now includes Bentley's CONNECT Advisor, which allows you to search Communities, LEARN content, online videos, and more from one convenient window.

**Related Links**

- **Bentley CONNECT Advisor Dialog** (on page 2279)

**RR 21.00.00-1.3 64-bit Analysis Engine**

The STAAD analysis and design engine now utilizes a 64-bit solver.

This allows you to analyze larger, more complex models with more objects and load cases. The analysis engine is faster for models of all sizes.

**RR 21.00.00-1.4 Web Help**

The product documentation is now delivered as a browser-based "Web Help." The Web help opens in your default browser.

Refer to **GS. Using Online Help** (on page 24) for additional details on using Web Help.

**RR 21.00.00-1.5 STAAD.Pro Script Editor**

A new macro editor is available to create and edit custom scripts to extend STAAD.Pro.

You can launch the new STAAD.Pro Script Editor application by selecting the **Macro Editor** tool in the **Developer** group on the **Utilities** ribbon tab.
RR 21.00.00-2 Features Affecting the Preprocessor

This section describes features that have been added that affect the preprocessor section of the program, also known as the Modeling Mode.

RR 21.00.00-2.1 STAAD.Pro Physical Modeler

You can use physical modeling concepts to generate structure geometry and reference load cases through the STAAD.Pro Physical Modeler interface.

You can launch STAAD.Pro Physical Modeler by selecting this as the modeling method when creating a new model on the Start page or by selecting the Physical Modeling workflow for an empty model.

Notes:
- Syncing physical model data will overwrite any model geometry created in the analytical modeling interfaces (i.e., traditional STAAD.Pro user interface or STAAD.Pro Editor).
- You cannot edit geometry, materials, reference loads, etc. created in the physical modeling interface from either the STAAD.Pro user interface or the STAAD.Pro Editor. Physical models must be edited in STAAD.Pro Physical Modeler.
- It is recommended that you begin a physical model by selecting the Physical Modeler from the New Model Wizard.

The STAAD.Pro tutorials (on page 373) have been re-written to incorporate the physical modeling workflow. It is recommended that even experienced STAAD users work through these tutorials to familiarize themselves with STAAD.Pro Physical Modeler.

Similarly, take some time to review the Quick Overview and Getting Started sections of the STAAD.Pro Physical Modeler help as they describe many of the differences in working in STAAD.Pro Physical Modeler versus the traditional STAAD.Pro interface and STAAD input files.

RR 21.00.00-2.2 NBCC 2010 Seismic Load

The program can now generate seismic loads per the National Building Code of Canada, 2010 edition.

Both the equivalent lateral force method and seismic response spectra are now supported per the NRC 2010.

Related Links
- TR.31.2.4 Canadian Seismic Code (NRC) - 2010 (on page 2556)
- TR.32.10.1.3 Response Spectrum Specification per NRC 2010 (on page 2702)

RR 21.00.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.
RR 21.00.00-3.1 NZS3404:1997 Steel Design

The program can now perform steel design per NZS3404: 1997 New Zealand Standard for Steel Structures, Parts 1 & 2, including Amendments 1 & 2.

Refer to D1.1.A. New Zealand Codes - Steel Design per NZS 3404-1997 (on page 2021) for details on this code implementation.

RR 21.00.00-3.2 ACI 318-14 Technical Preview

The program can now design concrete columns and beams per ACI 318-14 Building Code Requirements for Structural Concrete.

Note: This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

Related Links

- D1.F.1 Design Operations (on page 1479)

RR 21.00.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 21.00.00-4.1 RAM Connection Workflow Update

RAM Connection CONNECT Edition is now supported in STAAD.Pro CONNECT Edition.

- Connection Design per the IS800-2007(Indian) code

STAAD.Pro V8i

The Software Release Reports for STAAD.Pro V8i contain detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 03. These documents should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

V8i (SELECTseries 6)

The Software Release Report for STAAD.Pro V8i (SELECTseries 6) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 5) (release 20.07.10) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-11.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.
AD.2007-11.1.1 Mode Bar Reorganization

The modes in STAAD.Pro have been reorganized to reflect a linear workflow through modeling, postprocessing, and design stages of a project.

Note: This feature has been replaced by the Workflows feature with the CONNECT Edition Edition of STAAD.Pro.

Additionally, the tabs are color coded by group:

- Blue: preprocessing and modeling
- Light green: postprocessing
- Dark green: foundation design
- Yellow: steel design
- Red: concrete design
- Magenta: earthquake checks

AD.2007-11.1.2 Advanced Analysis Engine Enhancements

The advanced analysis engine has been completely reformatted with new routines to provide even faster methods to build and solve the stiffness matrix.

Arnoldi/Lanczos Eigen Method

In addition to the standard subspace iteration method for eigen solution, the Advanced Math Solver can use the Arnoldi/Lanczos method. For large scale eigen value problems, the Arnoldi method is very efficient.

Load Dependant Ritz Vectors

Ritz vector analysis can be used for dynamically loaded structures to more efficiently evaluate the relevant modes. It can require significantly less computational effort to evaluate a large structure when compared to natural free-vibration methods.
Autoshifting of Eigen Vectors

For subspace iteration and the Arnoldi/Lanczos methods, an autoshift option can be used to reduce memory demand for large structures.

Refer to G.17.3.1 Solution of the Eigenproblem (on page 2363) for details.

AD.2007-11.1.3* CONNECT Enabled

STAAD.Pro is now CONNECT Enabled. This means that you can associated your STAAD.Pro models with ProjectWise Projects from within the application to track application usage per project.

You must sign in with your Bentley CONNECT account. The Bentley CONNECTION Client is used to sign in and the status is displayed in the user icon in the top-right corner of the STAAD.Pro application window.

A new menu, Cloud Services, has been added which contains ProjectWise Project features, links to cloud portals, and a link to AD.2007-10.1.3* Scenario Services (on page 119).

Once you start a new model or open an existing model that is not yet associated with a ProjectWise Project, you will be prompted to assign a ProjectWise Project to your model. A list of all registered projects in your organization is displayed. You can also register new projects (Only users with Admin/Co-admin roles can register a project). STAAD.Pro will display the ProjectWise Project name in the Job Info dialog as well as in the window title bar.

Tip: As of CONNECT Edition, the Cloud Services tools can be found on the File ribbon tab in the Backstage.

For additional information on CONNECTED Projects, please see http://www.bentley.com/en-US/Promo/CONNECT/why+connect.htm.

Refer to Cloud Services tab (on page 2878) for details.

AD.2007-11.1.4† RAM Connection CONNECT Edition v11 Support

STAAD.Pro now supports RAM Connection CONNECT Edition (v11.0) and in the Connection Design workflow.
• STAAD.Pro is now compatible with versions of RAM Connection CONNECT Edition up through release 11.0. The following design codes are supported:
  
  • AISC 360-10 (LRFD and ASD)
  • GB 50017-2003 (China)
  • EN 1993-1-8 (Eurocode 3)
  
• You can export the Connection Design reports (on page 971) directly to a Microsoft® Office Word® document from the STAAD.Pro interface.

AD.2007-11.1.5† Connection Tags Enhancements

You can now assign user-defined equations for checking connection tags in STAAD.Pro. The XML Schema for the connection tag files has been updated. See “Connection Tags XML File Schema” (on page 986) in the User Interface help for details on the schema, as well as on using wildcards and created user-defined equations.

AD.2007-11.1.6† Add Member Enhancements

The Add Member functionality in the graphical user interface has been enhanced to facility the saving, loading, and managing of user-defined attribute sets.

Select Geometry > Add Beam > Set New Member Attributes to open the Define Member Attributes dialog (on page 2905). Here you can define the default attributes for various predefined attribute sets, as well as create user-defined attribute sets.

AD.2007-11.1.7† ISM Integration

STAAD.Pro is now capable of transferring data to and from v5 of Bentley's Integrated Structural Modeling technology by means of the StructLink utility.

ISM v5 includes support information, ProjectWise integration, and substructures. Refer to the ISM documentation for additional details.

AD.2007-11.2 Features Affecting the Preprocessor

This section describes features that have been added that affect the preprocessor section of the program, also known as the Modeling Mode.

AD.2007-11.2.1 Building Planner

This modeling tool allows you to rapidly generate concrete building structure models for analysis and design in STAAD.Pro and detailing in the RC Designer mode.

PlanWin and FrameWin have now been directly integrated into the STAAD.Pro user interface as the Building Planner mode.

Building Planner is primarily intended for design of concrete building structures for the Indian code.

Use of Building Planner is a separate license, which can be selected in the License Configuration section on the STAAD.Pro start page.

Refer to the Building Planner mode (on page 895) for details.
AD.2007-11.2.2 STAAD Editor

A new STAAD editor has been added which has advanced features for manually editing your STAAD input files, including IntelliSense.

Model Navigation

Associated commands are "blocked" together in the editor such that can be managed easier. Larger blocks can be collapsed to show only the first line/command, making it easier to navigate through large models.

A document outline of the model is also in a separate pane. You can double-click any entry in the outline to jump to that point in the input file.

IntelliSense

Intelligent code completion is now integrated into the STAAD editor. This allows context-aware code completion suggestions when you type in the editor.

Hover your mouse pointer over the suggested keywords to view tool tips for each.

Section profiles in standard tables are also linked to the editor. When entering a table shape, simply select it from the list of shapes in the drop-down list.

Enhanced Help

Press <F1> to view the help topic related to the command keyword from the Technical Reference help.

Refer to the STAAD Editor (on page 2251) help for details.

AD.2007-11.3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

Note: Items labeled with an obelisk (i.e., dagger) (†) were added or updated in the Service Pack 2 release of V8i (SELECTseries 6).

Note: Items labeled with a double obelisk (i.e., dagger) (††) were added or updated in the Service Pack 3 release of V8i (SELECTseries 6).
AD.2007-11.3.1 ACI 318-11 Concrete Design

Concrete design per the 2011 edition of the ACI 318 specification (ACI 318-11) is now available in STAAD.Pro. Several new parameters have been introduced to accommodate design of slender columns using the moment magnification procedure for sway and non-sway frames.

Refer to D1.F.5.6.1 Slenderness Effects and Analysis Consideration (on page 1521) for details.

To design concrete columns per ACI 318-11

The general outline of this procedure applies to designing all concrete members per ACI 318-11, but the specific parameters mentioned pertain only to the design of columns using the moment magnification method.

1. In the Modeling mode, select the Design | Concrete tab.

   The Concrete Design - Whole Structure dialog opens.

2. In the Current Code drop-down list, select ACI 318-11.

   This will insert the following commands into the STAAD.Pro input file:
   
   ```
   CODE ACI
   ```

3. Click Define Parameters.

   The Design Parameters dialog opens.

4. Select the MMAG parameter and type 0 for the option and then click Add.

   This instructs the program to use the detailed calculations considering sway or non-sway frames.

5. Select the SWY parameter and select the appropriate bracing condition for the frame containing the columns to be designed and then click Add.

6. (Optional) Set values for BDY, BDZ, SKY, SKZ, SLY, SLZ, SQY, SQZ, SWY, and TRN parameters as necessary (i.e., when the defaults to not match the column conditions).

7. Close the Design Parameters dialog and assign the parameters to the appropriate members.

   Tip: The parameters mentioned here are only applicable to columns. They will be ignored for beam members.

AD.2007-11.3.2 CAN/CSA S16-14 Steel Design

Steel design per the 2014 edition of the Canadian S16 code, Limit States Design of Steel Structures, is now available.

The performance of designs per CAN/CSA S16-09 has also been improved. The changes in code checks from the 09 and 14 edition of the codes are noted in D4.E.6.1 Members Subject to Axial Forces (on page 1730).

To specify a design using CAN/CSA-S16-14

1. Select the Design | Steel page.

   The Steel Design - Whole Structure dialog opens.

2. In the Current Code list, select the CAN/CSA-S16-14.

3. Click Define Parameters list.

   The Define Parameters dialog opens.

4. Specify one or more design parameters as necessary.
Note: Refer to the descriptions in the dialog or to D4.E.7 Design Parameters (on page 1736) for definitions of the design parameters and default values used.

5. Click Add.

This will insert the following commands into the STAAD input file:

```
CODE CANADIAN
```

Tip: To use the older edition of the Canadian steel design code, the command CODE CANADIAN 2009 is required.

AD.2007-11.3.3 SANS 10162-1:2011 Steel Design

Steel design per the South African SANS 10162-1:2011 code is now available.

Specifying a design using SANS 10162-1:2011

1. Select the Design | Steel page.
   
   The Steel Design - Whole Structure dialog opens.

2. In the Current Code list, select the SANS10162-1:2011.

3. Click Define Parameters list.

   The Define Parameters dialog opens.

4. Specify one or more design parameters as necessary.

   Note: Refer to the descriptions in the dialog or to D14.C.7 Design Parameters (on page 2207) for definitions of the design parameters and default values used.

5. Click Add.

   This will insert the following commands into the STAAD input file:

   ```
   CODE SANS10162-1:2011
   ```

   Tip: Alternately, SANS10162 will also use the latest edition. To use the older edition of the South African steel design code, SANS10162-1: 1993 is required.

Refer to D14.C. South African Codes - Steel Design Per SANS 10162-1:2011 (on page 2200) for details.

AD.2007-11.3.4 SP 63.13330-2012 Concrete Design

Concrete design per the Russian code SP 63.13330-2012 is now supported.

Refer to D13.D. Russian Codes - Concrete Design Per SP 63.1330.2012 (on page 2164) help for details.

AD.2007-11.3.5 AISC 360-05/10 Metric Steel Design

Steel design in metric per the AISC 360-05/10 code is now supported.

When you select metric units in STAAD.Pro, the full user interface (including section properties, results, etc.) displays the correct metric units for AISC 360-05/10 steel design. Additionally, the results are also fully in metric units.
AD.2007-11.3.6 AISC 360-05/10 Tapered Member Design

The design of tapered square and round tube shapes as well as tapered I-shaped members is now available per the AISC 360-10 code.

The design of tapered members was previously available under the AISC 360-05 code, but the performance of design per this code has also been enhanced. Refer to D1.A.5.7 Design of Web-Tapered Members (on page 1377) for details.

AD.2007-11.3.7 AISC 341-05/10 Seismic Provision Checks

Seismic provisions of AISC 341-05/10 can now be checked as part of an AISC 360-05/10 member design.

Several additional parameters have been added to the AISC 360 unified codes (both 2005 and 2010 editions) to check the additional seismic provisions of AISC 341. Refer to D1.A.9 Seismic Provision Checking per AISC 341 (on page 1388) for details.

AD.2007-11.3.8 IBC 2012 / ASCE 7-10 Seismic Loads

Automatically generated equivalent lateral seismic loads and response spectra per IBC 2012 / ASCE 7-10 are now available in STAAD.Pro.

The load parameters are assigned using the User Interface or can be manually entered in the STAAD editor. Automatic load combinations using the tables from IBC 2012 can be added using the Auto Load Combination dialog (on page 3033), as well.

To specify an IBC 2012/ASCE 7-10 static seismic load

1. Either:
   - Select Commands > Loading > Definitions > Seismic Load > IBC 2012 Load
   or
   - Select the Definitions section in the Load & Definitions dialog and then click Add.

   The Create New Definitions / Load Cases / Load Items dialog opens with the Add New Seismic Definitions dialog (on page 3044) tab selected.
2. Select IBC 2012 ASCE 7-10 in the Type drop-down list.

3. (Optional) Select the Include Accidental Load checkbox to consider accidental torsion.

4. Either:
   - Type a ZIP code
   - or
   - Type Latitude and Longitude values
   - or
   - Type Ss and S1 mapped values.

5. Specify the required seismic load parameters:
   - Type long-period transition time, TL, value in the indicated units.
   - Type an Importance factor (I) value,
   - Type Response Modification Factor X (RX) and Response Modification Factor Z (RZ) values, respectively.
   - Select the Site class (SCL) from the drop-down list.

6. (Optional) Specify the following optional seismic parameters as necessary:
a. Type short-period site coefficient, $F_a$, and long-period site coefficient, $F_v$, values, respectively.

**Note:** These values are required if Site class (SCL) of 6 is selected (corresponds to IBC 2012 site class of F).

b. Type a CT value used to calculate the time period.

c. Type the Period in X Direction (PX) and Period in Z Direction (PZ) values, respectively. Otherwise these values are calculated from the code.

d. Type an exponent value, $x$, for use in ASCE 7-10 equation 12.8-7.

7. Click Add.

The IBC 2012 parameters are added to the Seismic Definitions section of the Load & Definition dialog.

Refer to TR.31.2.13 IBC 2012 Seismic Load Definition (on page 2596) for details.

**AD.2007-11.3.9 Eurocode 3 Steel Grades**

SGR values now are aligned with Table 3.1: EN 1993-1-1: 2005.

SGR 3, 4 have been changed to represent S450 and S275 N/NL grades respectively.

**Related Links**

- [DS.C.6 Design Parameters](on page 1776)

**AD.2007-11.3.10 AIJ 2002 and 2005 New Design Parameters**

The following design parameters have been added or updated in the AIJ 2002 and 2005 design codes in STAAD.Pro.

- The MISES parameter now has an option to ignore torsion in the calculation of the Von Mises stresses.
- A new parameter, MBG, has been added to ignore the web of H-shape, I-shape, and channel sections in calculating the section modulus about the local Z axis.
- A new parameter, YNG, has been added to the AIJ 2002 implementation to instruct the design engine how to evaluate Young's Modulus, $E$.

Refer to the codes in [D9. Japanese Codes](on page 1969) help for details.

**AD.2007-11.3.11 Steel Design Code Performance Improvements**

Performance of steel design per CSA S16-509 and IS800:2007 codes has been improved.

**AD.2007-11.3.12† NRC 2005 Seismic Loads Updates and Additions**

Seismic loads per the National Building Code of Canada's NRC 2005 have been updated and expanded to include torsion in the static load equivalent method as well as response spectrum loading.

**AD.2007-11.3.13† IS 800 Design of Additional Steel Shapes**

The design of both double-channel face-to-face and I-sections with cover plates is now available per the IS 800 2007 specification.

Refer to “Assigning Properties from Steel Tables” (on page 2461) ” for details on assigning double-channel front-to-front sections (FR) and I-sections with top and bottom plates (TB).
AD.2007-11.3.14† AISC 360 Design of I-Sections with Cover Plates

The design of I-sections with cover plates is now available per the AISC 360 2005 and 2010 specifications. Refer to "Design of I-Section with Cover Plates" (on page 2461) for details on the assumptions and limit states used in the design of these sections. Refer to "Assigning Properties from Steel Tables" (on page 2461) for details on assigning I-sections with top and bottom plates (TB).

AD.2007-11.3.15† IS 800 Design Updates

The following additions and enhancements were added to steel design per the IS 800 2007 specification:

- Flexural-torsional buckling of single angles per Cl. 7.5.1.2.
- Minimum web thickness is checked against the serviceability requirements in Cl. 8.6.1.1 and the compression flange buckling requirement of Cl. 8.6.1.2.
- The various sections of Amendment-1 to the IS:800-2007 Code, dated January 2012, are implemented in STAAD.Pro as applicable.

AD.2007-11.3.16† Floor and One-Way Load Panel Information Printing

The program can now include floor load and one-way load panel information in the output (.ANL) file or in an external text file. Refer to the Create New Load Items dialog Floor Load tab (on page 3012) in the User Interface help for information on using this feature through the graphical user interface. Refer to TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for information on the command input for this feature.

V8i (SELECTseries 5)

The Software Release Report for STAAD.Pro V8i (SELECTseries 5) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 4) (release 20.07.09). This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-10.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.
**AD.2007-10.1.1 Bentley Trust Licensing**

STAAD.Pro now utilizes Bentley’s Trust Licensing program.

This allows you or your organization the flexibility to utilize the program as needed without interruption, regardless of the availability of licenses. The usage data is collected periodically for billing at a later time for your convenience.

Refer to the *Getting Started & Troubleshooting Guide* (installed with STAAD.Pro) for additional information on license setup for STAAD.Pro.

Visit [Bentley.com](http://Bentley.com) for more information on Bentley’s licensing solutions.

**AD.2007-10.1.2 Brazilian Steel Databases**

Brazilian steel section databases have been added to the section profile tables included in STAAD.Pro.

To select a Brazilian steel profile shape

1. Select the **General | Property** page.
   
   The **Properties - Whole Structure** dialog opens.
2. Click **Section Database** on the **Properties - Whole Structure** dialog.
   
   The **Section Profile Table** dialog opens.
3. On the Steel tab, expand the **Brazilian** entry to select a shape class.
4. Select a shape and specify any shape parameters in the dialog.
5. Click **Add**.
   
   The section is added to the list on the **Properties - Whole Structure** dialog Section tab.

**AD.2007-10.1.3* Scenario Services**

STAAD.Pro projects can be run in Bentley’s Scenario Services (formerly ProjectWise Scenario Services or Bentley CONNECT Scenario Services) server with a SELECT account. This allows you to run large models or complex, nonlinear analyses in the “cloud,” leveraging extensive computing power. Additionally, STAAD.Pro models can be added Multiple Discipline Optimization (MDO) projects.
Refer to Cloud Services tab (on page 2878) for additional information on using this feature.

AD.2007-10.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

![STAAD Analysis and Design](image)

**Note:** Items labeled with an asterisk (*) were added or updated in the QA&R release of V8i (SELECTseries 5) (Build 20.07.10.XX).

AD.2007-10.2.1 Advanced Cable Analysis

An advanced, nonlinear cable analysis has been added to STAAD.Pro. The nonlinear nature of the static solver can utilize either the Newton-Rapshon method or a modified Newton-Raphson method for reduced computational effort. The cables are considered as geometrically nonlinear.

Refer to the following procedures:

- **M. To add a member specification** (on page 797)
- **A. To specify a nonlinear cable analysis** (on page 931)
- **A. To specify post-analysis print commands** (on page 939)

Refer to G.17.2.8 Advanced Nonlinear Cable Analysis (on page 2358) for details on the analysis methodology used by STAAD.Pro.

AD.2007-10.2.2 Colombian Seismic Code

Static lateral forces for seismic loads based on Reglamento Colombiano Sismo Resistente (NSR-10) (2010 edition of Colombian seismic code)

**Note:** Refer to TR.31.2.7 Colombian NSR-10 Seismic Load (on page 2573) for details on the command parameters.

AD.2007-10.2.3 Canadian Steel Code Update

Steel design CAN/CSA-S16-09, Design of Steel Structures, is now available in STAAD.Pro.

Refer to D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14 (on page 1728) for additional information.

To specify a design using CAN/CSA-S16-09

1. Select the Design | Steel page.
   - The Steel Design - Whole Structure dialog opens.
2. In the Current Code list, select the CAN/CSA-S16-09.
3. Click Define Parameters list. The Define Parameters dialog opens.
4. Specify one or more design parameters as necessary.

Note: Refer to the descriptions in the dialog or to D4.E.7 Design Parameters (on page 1736) for definitions of the design parameters and default values used.

5. Click Add.

This will insert the following commands into the STAAD input file:

```
CODE CANADIAN
```

Tip: To use the older edition of the Canadian steel design code, the command `CODE CANADIAN 2001` is required.

AD.2007-10.2.4 Eurocode 3 National Annexes

Two additional country’s National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Germany and Sweden.

As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.

```
AD.2007-10.2.4.1 German National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the German NA to EC3, use the following procedure.
```

When the German National Annex to EC3 is used for design, the output section title is revised to include the German National Annex (National Annex to DIN EN 1993-1-1:2005). Additionally, the partial safety factors used are included in the output and are specified in the German NA.

For additional information, please refer to D5.D.12 German National Annex to EC3 (on page 1849) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754).

1. In the Modeling mode, select the Design | Steel tab.
   The Steel Design - Whole Structure dialog opens.
3. Click Define Parameters.
   The Design Parameters dialog opens.
4. Select the NA parameter in the list box.
5. Select the (10) Germany in the Select National Annex drop-down list.
6. Click Add.

   This will insert the following commands into the STAAD.Pro input file:

   CODE EN 1993-1-1:2005
   NA 10

AD.2007-10.2.4.2 Swedish National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Swedish NA to EC3, use the following procedure.

The Swedish National Annex document referred to is “BFS EN 1993-1-1:2005.”

When the Swedish National Annex to EC3 is used for design, the output section title is revised to include the Swedish National Annex (National Annex to EN 1993-1-1:2005). Additionally, the partial safety factors used are included in the output and are specified in the Swedish NA.

For additional information, please refer to D5.D.13 Swedish National Annex to EC3 (on page 1853) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754).

1. In the Modeling mode, select the Design | Steel tab.
   The Steel Design - Whole Structure dialog opens.
3. Click Define Parameters.
   The Design Parameters dialog opens.
4. Select the NA parameter in the list box.
5. Select the (11) Sweden in the Select National Annex drop-down list.
6. Click Add.

   This will insert the following commands into the STAAD.Pro input file:

   CODE EN 1993-1-1:2005
   NA 11

AD.2007-10.2.5 AISC 360-10 Torsion Design

STAAD.Pro can now design open shapes for stresses due to torsion per AISC Design Guide #9, Torsional Analysis of Structural Steel Members.
A new TORSION parameter for the AISC 360-10 steel design code which can be set to instruct the program to perform torsion checks per this guide.

Refer to D1.A.5.6 Design for Torsion (on page 1374) for details on the checks performed.

**To include torsion checks per Design Guide #9**

1. In the Modeling mode, select the Design | Steel tab.
   
   The Steel Design - Whole Structure dialog opens.

2. In the Current Code drop-down list, select AISC 360-10.

3. Click Define Parameters.
   
   The Design Parameters dialog opens.

4. Select the TORSION parameter and select 1 for the option.

5. (Optional) Select the TRACK parameter and select 3 for the option.
   
   This will include detailed output for the Design Guide #9 checks in the output.

6. Click Add.
   
   This will insert the following commands into the STAAD.Pro input file:

   ```plaintext
   CODE AISC UNIFIED 2010
   TORSION 1
   TRACK 3
   ```

7. For the TORSION parameter (and for the optional TRACK parameter), you must specify a member list using the assignment tools in the Steel Design - Whole Structure dialog.

**AD.2007-10.2.6 Missing Mass**

Missing mass can now be specified for time history load definitions.

Missing mass and rigid body modes are reported in the output and can be selected in the post-processing mode drop-down (* for mm & + for rb)

![Figure 6: Missing mass modes are marked with an (*) in the Modes drop-down list](image)
Rigid body modes are used internally to establish relative and absolute structure displacement, particularly for structures supported on relatively weak springs. Refer to TR.31.4 Definition of Time History Load (on page 2630) and TR.34.2 Modal Calculation Command (on page 2791) for additional information.

AD.2007-10.2.7* Wind Loads per Russian Design Code SP 20.13330.2011
Refer to TR.31.3 Definition of Wind Load (on page 2623) for additional information.

Refer to TR.32.10.1.12 Response Spectrum Specification per SP 14.13330.2011 (on page 2764) for additional information.

AD.2007-10.2.9* Steel Design per Russian Design Code SP 16.13330.2011
Steel member design per SP 16.13330.2011, Steel Structures, is now available in STAAD.Pro. Refer to D13.C. Russian Codes - Steel Design Per SP 16.13330.2011 (on page 2152) for additional information.

Specifying a design using SP 16.13330.2011

1. Select the Design | Steel page.
   The Steel Design - Whole Structure dialog opens.
3. Click Define Parameters list.
   The Define Parameters dialog opens.
4. Specify one or more design parameters as necessary.
   Note: Refer to the descriptions in the dialog or to D13.C.4 Design Parameters (on page 2156) for definitions of the design parameters and default values used.
5. Click Add.
   This will insert the following commands into the STAAD input file:
   CODE RUSSIAN
   Tip: To use the older edition of the Russian steel design code, the command CODE RUSSIAN 1990 is required.
6. Assign the code parameters to members as necessary.

AD.2007-10.3 Features Affecting Post Processing
The following new feature has been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

Note: Items labeled with an asterisk (*) were added or updated in the QA&R release of V8i (SELECTseries 5) (Build 20.07.10.XX).
AD.2007-10.3.1 RAM Connection Mode Update

RAM Connection V8i features are now supported in STAAD.Pro V8i (SELECTseries 5) RAM Connection mode.

- Now compatible with versions of RAM Connection V8i up through release 8.0 (SELECTseries 4).
- New templates:
  - US: MEP Knee BCF, Moment End Plate BCF, Moment End Plate BCW, Moment End Plate BS Apex, and Moment End Plate BS
  - UK: Bolted End Plate BS Apex
- New sections added to the RAM Connection database for connection design:
  - Canadian WS, MS, S, HP, Angel, Channel, MC Channel, and HSS Round shapes
  - Brazilian I, Angle, and Channel shapes

AD.2007-10.3.2* CAN/CSA-A23.3-10 in RC Designer

The RC Designer module (Concrete Design mode) can now be used to design beams and columns per CAN/CSA A23.3-10.

- Beam (on page 1021) and Column (on page 1024) design per CAN/CSA A23.3-10 – The design of concrete columns and beams per the Canadian concrete code is now supported.

V8i (SELECTseries 4)

The Software Release Report for STAAD.Pro V8i (SELECTseries 4) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-09.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

Note: Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 4) (Build 20.07.09.22).

AD.2007-09.1.1 I-Section with Different Flange Shapes

User-provided steel table Wide Flange sections can now have different flange sizes on top and bottom.
Previously, the sections were required to be doubly symmetric. Now, you may specify a different bottom flange size.

To add a UPT I-Section with different flange sizes

1. (Optional) Select Tools > Select Input Units. Select appropriate units for length and click OK.
2. Select Tools > Create User Table.
   If no User Defined Table exists, you will be prompted to create one.
   The Create User Provided Table dialog opens.
3. Click New Table.
   The Select Section Type dialog opens.
4. In the Select Section Type list, select WIDE FLANGE and then click OK.
   Tip: You can also use this dialog to save the User Provided Table as a separate file.
   The dialog closes and the Select Existing Table list in the Create User Provided Table dialog now has at least one entry.
5. Click Add New Property.
   The Wide Flange dialog opens.
6. Type dimensions in the \textbf{D}, \textbf{TF}, \textbf{WF}, \textbf{TW}, \textbf{TF1}, and, \textbf{WF1} fields to define the section.

Two new fields— \textbf{TF1} and \textbf{WF1} —are used to specify different dimensions for the bottom flange. If these fields are left empty, the values for the top flange are used (doubly symmetric section).

\textbf{Note:} Refer to \textit{TR.19.1 Wide Flange} (on page 2449) for detailed descriptions on properties of Wide Flange sections in User Steel tables.

7. Click \textbf{Calculate}.

The derivative cross section properties are calculated.

8. Click \textbf{OK}.
The dialog closes and the section is added to the **Table Data** list in the **Create User Provided Table** dialog.

**AD.2007-09.1.2 I-Section with Flange Plates or Composite Slab**

User-provided steel table Wide Flange sections can now have an additional bottom flange plate or composite concrete on the top flange.

These options are available in the **Wide Flange** dialog in the user interface. The additional flange plates or composite slab data is then used along with the base section properties.

**To add a UPT I-Section with flange plates and composite slab**

1. (Optional) Select **Tools > Select Input Units**. Select appropriate units for length and click **OK**.
2. Select **Tools > Create User Table**.
   
   If no User Defined Table exists, you will be prompted to create one.

   The **Create User Provided Table** dialog opens.
3. Click **New Table**.
   
   The **Select Section Type** dialog opens.
4. In the **Select Section Type** list, select **WIDE FLANGE** and then click **OK**.

   **Tip:** You can also use this dialog to save the User Provided Table as a separate file.

   The dialog closes and the **Select Existing Table** list in the **Create User Provided Table** dialog now has at least one entry.
5. Click **Add New Property**.

   The **Wide Flange** dialog opens.
6. Add a composite flange:
   a. Check the option for **Additional Composite Flange**.
   b. Type dimensions in the **B(left), B(right), Thickness**, and, **Modular Ratio** fields to define the composite slab.

   **Note:** Refer to TR.19.1 Wide Flange (on page 2449) for detailed descriptions on properties of Wide Flange sections in User Steel tables.

7. To add a bottom cover plate, select the **Additional Bottom Steel Plate** option and type values for **B(left), B(right), and Thickness**.
Tip: Top cover plates cannot be added in association with a composite slab, only bottom cover plates.

8. Click Calculate.

The derivative cross section properties are calculated.

9. Click OK.

The dialog closes and the section is added to the Table Data list in the Create User Provided Table dialog.

AD.2007-09.1.3 ISM Integration

STAAD.Pro is now capable of transferring data to and from v3 of Bentley's Integrated Structural Modeling technology by means of the StructLink utility.

ISM v3 includes support for foundation information. STAAD Foundation Advanced is now ISM Enabled, so this allows for an improved method for transferring and updating support information for STAAD Foundation Advanced, as well as other ISM Enabled products using v3 of ISM, such as detailing products like ProConcrete.

AD.2007-09.1.4* Connection Tags

This feature is used to both create connection data within STAAD.Pro and to transfer it to other programs via the ISM link. This data can then be used in third-party programs, such as Tekla Structures®. The connections can then be checked against a defined capacity.

Connection tags are assigned and checked from within the STAAD.Pro User Interface by use of the right-click pop-up menu. When a beam member is selected (using the beam cursor), a sub-menu for Connection Tags is provided with tools for using connection tags.

Note: Refer to Connection Tags sub menu (on page 999) for additional information on using this feature.

Connection tags consist of two pieces of data:

i. A Connection Tags XML file, which contains the connection categories, tag names, and member end releases for the connection tag. Connection capacities are also specified for each combination of member and connecting member which may utilize a connection tag. Refer to Connection Tags XML File Schema (on page 986) for additional information on the required structure of this XML file.

ii. Assignments of connection tags to members are stored in the STAAD input file. Though this is done within the DEFINE MEMBER ATTRIBUTE command, it is strongly recommended that the user interface features be used to make connection tag assignments as these must utilize only the connection categories and tag names in the associated XML file. Refer to TR.29.2 Connection Tag Member Attribute (on page 2537) for additional information on this command.

AD.2007-09.2 Features Affecting the Pre-Processor

This section describes features that have been added that affect the pre-processor section of the program, also known as the Modeling Mode.

AD.2007-09.2.1 Print Center of Rigidity

The PRINT DIAPHRAGM CR command may be used to obtain a print-out of the center of rigidity and center of mass at each rigid diaphragm in the model.

The lateral force at each floor, as generated by earthquake and wind loading, acts at the center of rigidity of each floor which is modeled as rigid floor diaphragm. The center of mass of each floor is defined as the mean location
of the mass system of each floor. The mass of the floor is assumed to be concentrated at this point when the floor
is modeled as rigid diaphragm. The distance between these two is the lever arm for the natural torsion moment
for seismic loads when that option is used.

Related Links
- TR.42 Print Specifications (on page 2840)
- A. To output the center of rigidity (on page 940)

AD.2007-09.2.2 Load & Definition

A new feature has been added to include horizontal torsion for rigid floor diaphragms in the equivalent static
seismic analysis. This torsion—referred to as the natural torsion—accounts for the static eccentricity which is
the difference between center of mass and center of rigidity of a rigid floor diaphragm, to be used to multiply the
UBC, IBC, 1893, etc.

Refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773) for additional details.

Adding Natural Torsion factor to a static seismic load

The natural torsion factor is specified along with the accidental torsion factor.

1. Add a Seismic Load definition to your model.
2. Add a Load Case with the Loading Type of Seismic to your model.
3. Select the seismic load case in the Load & Definition dialog and click Add.
   The Add New: Load Items dialog opens.
4. Select the Seismic Loads tab.
5. Set the option to use the Multiplying factor for Natural Torsion Moment and type a value (less than or
equal to one).
6. Select the direction and specify the accidental torsion option as necessary.
7. Click Add.

**Related Links**
- **TR.32.12.2.1 Generation of Seismic Loads** (on page 2773)
- **TR.32.12.2.2 Generation of IS:1893 Seismic Load** (on page 2778)
- **M. To add a seismic load** (on page 842)

**AD.2007-09.2.3 Check for Soft Story**

The CHECK SOFT STORY option may be added to a FLOOR DIAPHRAGM command set to check for soft stories per either IS1893:2002 or ASCE7-95 codes.

Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command PRINT STORY STIFFNESS in the post analysis print commands section.

**Related Links**
- **Floor Diaphragm Options dialog** (on page 3079)
- **TR.28.2.1 Soft Story Checking** (on page 2529)
- **TR.28.2 Floor Diaphragm** (on page 2526)
- **TR.42 Print Specifications** (on page 2840)
- **A. To check for soft stories and seismic code irregularities** (on page 937)
AD.2007-09.2.4 Check Story Drift

The PRINT STORY DRIFT command can now be used to check the story drift against a code-specified maximum drift ratio.

Story drift is calculated as the relative horizontal displacement of two adjacent floors in a building. Inter-story drift can also be expressed as some factor times the story height. The allowable factor generally varies from one country code to another and may also vary depending on the type of loading. For example, in IS 1893: 2002 for seismic loading the allowable limit for inter-story drift is 0.004 times the story height whereas in IS 875 for wind loading it is 0.002 times the story height.

The drift at a particular story level in either lateral direction is calculated as the average of all the joint displacements present in that floor level. However if floor diaphragm is present, the drift is calculated at the center of mass (i.e., master joint) of the floor.

**Note:** Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command PRINT STORY STIFFNESS in the post analysis print commands section.

**Note:** For dynamic IS 1893: 2002 response spectrum, the story drift check is performed by adding a command line within the load case, rather than in the post-analysis print commands. Refer to TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2721) for additional information.

**Related Links**
- TR.42 Print Specifications (on page 2840)
- A. To check for inter-story drift (on page 941)

AD.2007-09.3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

**Note:** Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 4) (Build 20.07.09.22).

AD.2007-09.3.1 Steel Design per AISC 360-10

The design of steel sections per ANSI/AISC 360-10 Specification for Structural Steel Buildings (AISC 360-10) is now available in STAAD.Pro.

Refer to D1.A. American Codes - Steel Design per AISC 360 Unified Specification (on page 1366) for additional information.
To specify a design using AISC 360-10

1. Select the Design | Steel page.

   The Steel Design - Whole Structure dialog opens.

2. In the Current Code list, select the AISC 360-10.

3. Click Define Parameters list.

   The Define Parameters dialog opens.

4. Specify one or more design parameters as necessary.

5. Click Add.

   This will insert the following commands into the STAAD input file:

   ```
   CODE AISC UNIFIED 2010
   ```

AD.2007-09.3.2 Concrete Design per ACI 318-08

The design of concrete members per the ACI code in STAAD.Pro has been updated to the 2008 edition of the code.

A new parameter, LWF, has been added to account for the lightweight concrete reduction factor, λ, in equations 11-3, 11-4, and 11-5 of the code.

Refer to D1.F. American Codes - Concrete Design per ACI 318 (on page 1478) for additional information.

To specify a design using ACI 318-08 in batch mode

**Note:** Design per the 2008 edition of the ACI code is currently only available in the batch mode.

1. Select the Design | Concrete page.

   The Concrete Design - Whole Structure dialog opens.

2. In the Current Code list, select the ACI 318 2008.

3. Click Define Parameters list.

   The Define Parameters dialog opens.

4. Specify one or more design parameters as necessary.

5. Click Add.

   This will insert the following commands into the STAAD input file:

   ```
   CODE ACI
   ```

AD.2007-09.3.3 Malaysian National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Malaysian NA to EC3, use the following procedure.

The Malaysian National Annex document referred to is “MS EN 1993-1-1:2005.”

When the Malaysian National Annex to EC3 is used for design, the output section title is revised to include the Malaysian National Annex (National Annex to MS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Malaysian NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

**Note:** For additional information, please refer to D5.D.11 Malaysian National Annex to EC3 (on page 1844) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754).
1. In the Modeling mode, select the Design | Steel tab.  
The Steel Design - Whole Structure dialog box opens.
3. Click Define Parameters…  
The Design Parameters dialog box opens.
4. Select the NA parameter in the list box.
5. Select the option for (9) Malaysia.
6. Click Add.

This will insert the following commands into the STAAD input file:

<table>
<thead>
<tr>
<th>CODE</th>
<th>EN 1993-1-1:2005</th>
</tr>
</thead>
<tbody>
<tr>
<td>NA</td>
<td>9</td>
</tr>
</tbody>
</table>

AD.2007-09.3.4 Star Angle Design per IS-800

The design of "star" angle arrangements per IS 800:2007 have been implemented in STAAD.Pro to for members subjected to axial force only. Such star angles are often used for the legs of transmission or communication towers and as bracing members in industrial buildings.

Refer to D8.E. Indian Codes - Steel Design per IS 800 - 2007 (on page 1942) for details on Limit States and Member Property Specifications in IS 800:2007 design.

Notes

- The design of star angles is only supported for IS 800:2007.
- Members must be declared as a TRUSS member (axial only). An error will be reported in the output if any other member specification is used.
- It is assumed that the star angle arrangement is a welded shape. Plated shapes are not accounted for in the program.

To specify a star angle arrangement for IS 800:2007 design

Refer to TR.20.1 Assigning Properties from Steel Tables (on page 2461) for additional information on assigning properties from steel tables.

1. Select Commands > Member Property > Steel Table > Indian.

The Section Profile Tables dialog opens to the Indian steel table.
2. Select the Angle tab.
3. Chose an angle section from the Select Angle list.
4. Select the SA (Star angle arrangement, Double Angle) Type Specification.
5. Click Add.

The section is added to the model and is available for member assignment.

AD.2007-09.3.5 IS 1893 (Part 4) 2005 Seismic Loads

Seismic loads per IS 1893 (Part 4) 2005 for industrial and stack-like structures can be generated in STAAD.Pro. These loads have been incorporated into the existing implementation of IS 1893 (Part 1) 2002 loads.

To add a seismic load per IS 1893 (Part 4) 2005

The Create New Definitions / Load Cases / Load Items dialog opens to the Seismic Parameters tab (only tab visible in this interface) and the IS 1893 - 2002/2005 code is selected.

2. Set the option for Include 1893 Part 4.
3. Do either of the following:
   - Click Generate to open the IS:1893 Seismic Parameters dialog, or
   - Type values for the command parameters directly in the current dialog table.
4. Click Add.

**Note:** Refer to TR.31.2.9 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2576) for additional information.

**AD.2007-09.3.6* ABS/SRSS Combination**

A modification to the ABS (Absolute value) and SRSS (square root sum of the squares) results combinations methods.

Additional processing is now used so the resulting combination of results contain appropriate sign/direction as is required when used for design or code checking when used with the ASME NF or AISC 360 05/10 codes.

To instruct the analysis and design engine to automatically generate these load combinations for ABS or SRSS combinations, either:

- use the new, optional GENERATE parameter in the LOAD COMBINATION command, or

**Note:** Refer to TR.35 Load Combination Specification (on page 2791) for additional information.
- select the Generate Combination option in the Define Load Combinations dialog (on page 3003).

All of the permutations of positive or negative sign effects in each DOF (degree of freedom) are now considered by the analysis engine. That is, in each of the 6 DOF, both positive and negative sign are considered which results in 64 possible cases (i.e., $2^6$). As a result of this, the SRSS and ABS combinations are now presented as load envelopes (i.e., maximum and minimum values) at each member end. This is reflected in both reports and post-processing mode tables. Similarly, in the structure view in the post-processing mode, the force diagrams are now displayed graphically as an envelope rather than a single curve.

Output from the following commands displays pairs of results rows for a load at a joint (marked by an asterisk in the output):

- PRINT SUPPORT REACTION
- PRINT MEMBER FORCE
- PRINT SECTION FORCE
- PRINT PMEMB FORCE
- PRINT MEMBER STRESS

**V8i (SELECTseries 3)**

The Software Release Report for SELECTseries 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i SELECTseries 2 (release 20.07.07) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.
The Software Release Report for STAAD.Pro V8i SELECTseries 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i SELECTseries 2 (release 20.07.07) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

**AD.2007-08.1 Features Affecting the General Program**

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

**AD.2007-08.1.1 ISM Integration**

STAAD.Pro is now capable of transferring data to and from Bentley’s Integrated Structural Modeling technology by means of the StructLink utility.

**AD.2007-08.1.2 Export to SACS**

A new macro is included with STAAD.Pro which is used to export the current STAAD.Pro model to a SACS model, which can be opened in the SACS system Interactive Modeling program.

SACS (Structural Analysis Computer System) is a finite element structural analysis suite of programs for the offshore and civil engineering industries.

**Related Links**
- [I. To export to a SACS input file](on page 2289)

**AD.2007-08.1.3 European Cold Formed Sections per EN10219-2**

Sections defined in the publication, EN10219-2: 1997 *Cold formed welded structural hollow sections of non-alloy and fine grained structural steel. Part 2: Tolerances, dimensions and sectional properties* have been added to library of cross sections available in the program.

The following section profiles are included as separate tables in the program:

- Table 6: Circular Hollow Sections (CHS)
- Table 7: Square Hollow Sections (SHS)
- Table 8: Rectangular Hollow Sections (RHS)
To Specify a European cold formed section

1. Select the **General | Property** page.
   - The **Properties - Whole Structure** dialog box opens.
2. In the **Properties - Whole Structure** dialog box, click **Section Database**.
   - The **Section Profile Tables** dialog box opens.
3. Select the **Coldformed Steel** tab.
4. Select one of the tables available in the **European Cold Formed** section.
5. Select an entry from the **Select Profile** list.
6. Click **Add**.

   The section is now added to the Section tab in the **Properties - Whole Structure** dialog box and can be assigned to members.

AD.2007-08.1.4 Japanese JIS Hollow Sections

The catalog of available Japanese hot rolled steel sections has been expanded to include the hollow sections from JIS publications.

- JIS G3444:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for circular hollow sections and are classed as “General pipe sections”. The existing PIPE table has been updated to ensure that it is consistent with this table.
- JIS G3475:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for circular hollow sections and are classed as “Architectural pipe sections”. A new table for these Circular Hollow (CHS) will be added.
- JIS G3466:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for square and rectangular hollow sections. These will be added to the existing Japanese Sections database as two new tables: Rectangular Hollow (RHS) and Square Hollow (SHS)

Refer to [D9.C.4 Built-in Japanese Steel Section Library](on page 1989) for additional details on using the Built-in Japanese Steel Section Library.

To specify a Japanese hollow section

1. Select the **General | Property** page.
   - The **Properties - Whole Structure** dialog box opens.
2. In the **Properties - Whole Structure** dialog box, click **Section Database**.
   - The **Section Profile Tables** dialog box opens.
3. Select the **Steel** tab.
4. Select one of the tables available in the **Japanese** section: Pipe, Rectangular Hollow, Square Hollow, or Circular Hollow.
5. Select an entry from the **Select Profile** list.
6. Click **Add**.

   The section is now added to the Section tab in the **Properties - Whole Structure** dialog box and can be assigned to members.
AD.2007-08.2 Features Affecting the Pre-Processor

This section describes features that have been added that affect the pre-processor section of the program, also known as the Modeling Mode.

AD.2007-08.2.1 Wind Load Generation per ASCE 7-10

Wind loads intensity values may now be automatically generated per the 2010 edition of SEI/ASCE 7 using the ASCE-7: Wind Load dialog (on page 3039) in the user interface. This dialog allows you to generate a wind loading pattern based on the parameters used in this specification.

The primary change in wind load evaluation between the 2002 and 2010 editions of the ASCE 7 specification involve the method of calculating the force coefficient, \( C_f \), for solid freestanding walls and solid signs. The formula provided in the commentary related to Figure 6-20 in the specification is used:

\[
C_f = \frac{1.563 + 0.008542 \ln(x) - 0.06148 \cdot y + 0.009011 \cdot \ln(x)^2 - 0.2603 \cdot y^2 - 0.08393 \cdot y \ln(x)}{0.85}
\]

Where:

\[
x = \frac{B}{s} \\
y = \frac{s}{h}
\]

**To generate a wind load intensity per ASCE 7-10**

1. Add a wind load definition to the model.
2. Select this definition in the Load & Definition dialog box and click Add.
   
The Add New: Wind Definition dialog box opens.
3. On the Intensity tab, select the as Custom in the drop-down list and click Calculate as per ASCE-7.
   
The ASCE-7: Wind Load dialog box opens.
5. Specify or set the parameters needed to define the load.
   
   Click Apply prior to selecting a different dialog tab to update the dialog for the specified parameters.
6. Click OK.

AD.2007-08.2.2 Single Mass Model

A new load type, mass, is available for reference load cases. This is used to create a single mass model for all dynamic loads (i.e., seismic, response spectrum, time history, etc.). This load case can be used for seismic loads in lieu of a weight table, reducing repetitive data entry for analysis methods which would require the same data.

A new load type, mass, is available for reference load cases. This is used to create a single mass model for all dynamic loads (i.e., seismic, response spectrum, time history, etc.). This load case can be used for seismic loads in lieu of a weight table, reducing repetitive data entry for analysis methods which would require the same data.

Related Links

- TR.31.6 Defining Reference Load Types (on page 2642)
- M. To add a mass model reference load (on page 876)
- M. To add mass loads to the mass model reference load (on page 877)
AD.2007-08.2.3 Eurocode Load Combination Generator

A new macro has been included with the program to generate load combinations for the Strength limit state per Eurocode – Basis of structural design, BS EN 1990:2002+A1:2005.

The load combination generator is capable of creating load combinations per equations 6.10, 6.10a, or 6.10b found in Cl. 6.4.3.2.

These equations specify the following combinations of loads:

\[ \sum_{j \geq 1} \gamma G_{k,j} + \gamma P + \gamma Q,_{1} \Psi_{0,1} Q_{k,1} + \sum_{i > 1} \gamma Q,_{1} \Psi_{0,1} Q_{k,1} \]

Alternatively, for the strength limit state, the less favorable of equations 6.10a and 6.10b may be used:

\[ \sum_{j \geq 1} \xi j \gamma G_{k,j} + \gamma P + \gamma Q,_{1} \Psi_{0,1} Q_{k,1} + \sum_{i > 1} \gamma Q,_{1} \Psi_{0,1} Q_{k,1} \]  

(6.10a)

\[ \sum_{j \geq 1} \xi j \gamma G_{k,j} + \gamma P + \gamma Q,_{1} \Psi_{0,1} Q_{k,1} + \sum_{i > 1} \gamma Q,_{1} \Psi_{0,1} Q_{k,1} \]  

(6.10b)

where

- \( G_k \) = Permanent actions
- \( P \) = Prestress actions
- \( Q_k \) = Variable actions

**Note:** The effects in each of the above equations are always additive. If any effect is negative (that is, would reduce the final sum), its effect is taken as zero.

AD.2007-08.2.4 Rigid Floor Diaphragms

A new feature has been added to easily model a rigid floor diaphragm without the need to specify a master joint at each. When specified, this command directs the engine to perform the following:

a. calculate the center of mass for each rigid diaphragm (where master joint is to be located) considering the mass model of the structure. The mass must be modeled using the new mass reference load feature (on page 140).

b. create, internally, an analytical node at the center of mass location to be included during analysis (unless a master node is specified) if an existing analytical node exists at this point, then the existing joint is used in lieu of creating a new joint.

c. search all nodes available within a diaphragm and add them as slave nodes; with the master node located at the center of mass for the diaphragm (or at the specified master node)

**Related Links**

- **TR.28.2 Floor Diaphragm** (on page 2526)
- **Node Specification dialog** (on page 2960)
- **M. To assign nodes to a floor diaphragm** (on page 811)

AD.2007-08.3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.
AD.2007-08.3.1 API 2A WSD 21st Ed. Update

Joint checking for tubular members per the American Petroleum Institute 2A-WSD code has been updated to the 21st Edition (December 2000) of that code, including errata and supplements 1 to 3 (latest: Supplement 3 - March 2008). Additionally, the process of joint design has been simplified.

**Note:** Only simple joints and overlapping joints will be considered by the program. Other type such as grouted joints, joints with ring stiffeners etc are not be considered.

The clauses/sections in the API code that have been dealt with are:

- 4.2.1 Material strength
- 4.2.3 Minimum Capacity
- 4.2.4 Joint Classification
- 4.3 Simple joints
- 4.4 Overlapping joints

Refer to D1.J. American Codes - Steel Design per API 2A-WSD 2000 (on page 1562) for additional information on the methodology used.

**External Joint Data File**

In previous versions, the LEG design parameter was used to direct the program to generate a separate joint data file or to check a user-specified file. This file is now automatically generated or checked as needed.

As the API code allows for mixed joint types, the PUNCH column in the input file has been replaced with K, X, and Y columns which are used to designate fractional contributions of each joint class. Similarly, overlapping joints are indicated by using a negative value for the GAP between braces and the member number of the overlapping member in the OB column (replaces THETAT used in previous versions).

If the filename.PUN file is not detected in the same folder as the current STAAD input file (where "Filename" is the same as the .STD file), then the program assumes this is the initial joint design and this file is created. If, however, this file is detected, the program assumes that the that the joint design has been performed at least once and will use this file to perform the joint checks.

**To check tubular member joints per API**

The checking of joints is an iterative process done by means of an automatically generated text file.

1. Model a structure as you normally would.

   **Note:** Only circular pipe members are considered.

   **Tip:** Using TRUSS specifications helps to reduce analysis time.

2. In the **Steel Design - Whole Structure** dialog box, select **API** as the design code.
3. (Optional) Specify the factor of safety used for joint checks using the new FSJ design parameter.
4. (Optional) Specify all other necessary design parameters.
   The LEG parameter is no longer used to generate or check joint data files.
5. Specify CODE CHECK or SELECT MEMBER commands as needed and perform the analysis.
   If this the filename.PUN file is not detected in the same folder as the input file, the program assumes this is
   the first time the structure is being analyzed and generates this input file with default data for the detected
   joints. Each joint is assumed to be a Y joint by default.
6. Modify the default joint data in the filename.PUN as needed to describe the actual joint conditions.
   A text editor can be used to make changes to this file. Be sure to save changes once complete.
7. Re-analyze the STAAD input file.
   Joint check results follow the steel design output.
8. Repeat steps 3 through 7 as needed to make changes in the structure.

AD.2007-08.3.2 Shear Buckling per EC3

The design code checks performed per Eurocode 3 have been updated to include checks for shear buckling in I
Sections and PFC Sections. Eurocode 3 – Part 1 (EN 1993-1-1:2005) states in Cl.6.2.6 that the checks for Shear
buckling are to be based on the procedure in EN 1993-1-5. STAAD.Pro performs checks based on the methods
Section 5 of EN 1993-1-5:2006.

In the case of an unstiffened web, the program will check the unstiffened web capacity. If the demand due to
applied loads is greater than this capacity, the program will calculate a suitable spacing for transverse stiffener
plates in order to meet the demand.

In the case of a web with transverse stiffeners, the program will check the capacity of the web considering the
provided stiffener spacing. The program will consider both the buckling capacity of the web as well as the flange.
If the demand due to applied loads is greater than this capacity, the program will calculate a reduced stiffener
spacing for transverse stiffener plates in order to meet the demand.

**Note:** Only transverse stiffeners are taken into account. The effect of any longitudinal stiffeners is ignored.

The distance between transverse stiffeners is specified by the STIFF parameter, in the current units of length. If
no value is specified, the program assumes a spacing equal to either the member length or depth of beam,
whichever is greater.

The output file provides recommendations on the evaluation of stiffeners (e.g., adding stiffeners at a spacing or
increasing the web thickness).

Refer to D5.C.5.3 Members Subject to Shear (on page 1760) for additional information.

AD.2007-08.3.3 IS800:2007 Working Stress Method

The IS:800-2007 Steel code was deviated in concept from its -1984 version (based on Working Stress Method)
and introduced the Limit State Method of Design. The entire 2007 version of the code is devoted to the Limit
State Method of Design, except Chapter 11. This Chapter comprises of a couple of pages and has the guideline for
the Design of Steel sections as per working stress method (WSM). The approach of this new working stress
method is different from its earlier version and utilizes the concept of Section Slenderness and Section
Classification.

The program now includes design per the Working Stress Method (WSM) methodology in addition to Limit State
method for design of steel structures per IS800:2007.
Some minor corrections to the Limit State Design option have also been made.

**To specify a design using IS 800:2007 Working Stress Method**

1. In the Modeling mode, select the Design | Steel tab.

   The Steel Design - Whole Structure dialog box opens.

2. In the Current Code drop-down menu, select IS800 2007 WSD.

3. Click Define Parameters...

   The Design Parameters dialog box opens.

4. Click Add.

   This will insert the following commands into the STAAD input file:

   ```
   CODE IS800 WSD
   ```

**AD.2007-08.3.4 Surface Element Selfweight**

A new surface load has been added to include the self weight of surface elements. This command can be used to calculate and include the weight of surface elements in the analysis of a structure.

**Related Links**

- TR.32.9.2 Surface Selfweight Load (on page 2686)
- M. To add a surface selfweight load (on page 832)

**AD.2007-08.3.5 Eurocode 3 National Annexes**

Two additional country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Singapore and Belgium. As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.
AD.2007-08.3.5.1 Belgian National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Belgian NA to EC3, use the following procedure.

The Belgian National Annex document referred to is “NBN EN 1993-1-1:2005.”

When the Belgian National Annex to EC3 is used for design, the output section title is revised to include the Belgian National Annex (National Annex to NBN-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Belgian NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

Note: For additional information, please refer to D5.D.10 Belgian National Annex to EC3 (on page 1838) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754).

1. In the Modeling mode, select the Design | Steel tab.
   The Steel Design - Whole Structure dialog box opens.
3. Click Define Parameters…
   The Design Parameters dialog box opens.
4. Select the NA parameter in the list box.
5. Select the option for (8) Belgium.
6. Click Add.

This will insert the following commands into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 8
```

AD.2007-08.3.5.2 Singaporean National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Belgium NA to EC3, use the following procedure.

The Singaporean National Annex document referred to is “SS EN 1993-1-1:2005.”

When the Singaporean National Annex to EC3 is used for design, the output section title is revised to include the Singaporean National Annex (National Annex to SS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Singaporean NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

Note: For additional information, please refer to D5.D.9 Singaporean National Annex to EC3 (on page 1831) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754).

1. In the Modeling mode, select the Design | Steel tab.
   The Steel Design - Whole Structure dialog box opens.
3. Click Define Parameters…
   The Design Parameters dialog box opens.
4. Select the NA parameter in the list box.
5. Select the option for (7) Singapore.
6. Click Add.
This will insert the following commands into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 7
```

**AD.2007-08.3.6 EC3 Slender Circular Hollow Sections**

Slender circular hollow (pipe) sections may now be designed per Eurocode 3 (EN 1993-1-6:2007).

Eurocode 3 – Part 1 (EN 1993-1-1:2005) –hereafter EC3-6– states in Cl. 6.2.2.5 (5) that the design of slender circular hollow sections is to be based on the procedure in EN 1993-1-6. EC3 6 deals with the design of shell structures. EC3-6 does not, however, specify additional or modified safety factors. Therefore, the default safety factors from EN 1993-1-1 are used.

**Note:** You can change these values through the GM0, GM1, and GM2 design parameters.

The program checks the plastic and buckling limit states for primary stresses based on the stress design method described in EC3-6.

Refer to [D5.C.5.6 Design of Slender pipe sections to EN 1993-1-6](on page 1771) for details on the methodology and calculations used in this design.

**AD.2007-08.3.7 User Defined Section for EC3**

The feature to design user-provided table (UPT) general sections has now been introduced steel members designed per Eurocode 3 (EN 1993-1-1:2005). However, rather than assuming that the section will behave like an I section, you are given the option of choosing the 'section-type' he would like to design the member for.

This is achieved through the introduction of a new design parameter, GST, that has the following values:

1. I-Section (Default)
2. Single Channel
3. Rectangular Hollow Section
4. Circular Hollow Section
5. Angle Section
6. Tee Section

Unless specified using the GST parameter, a general section will be assumed to be an I-Section.

**Note:** This parameter will be ignored if assigned to any section other than a General Section.

The design procedure will then account for the section type and proceed with the design as necessary. The output report will also indicate the section type considered for the design of the UPT section. The design output will indicate the section as follows:

```
*     1 ST IPE100 (UPT: DESIGNED AS I-SECTION)
   FAIL EC-6.3.2 LTB 8.591 1
   0.00 0.00 -31.25 2.50

<table>
<thead>
<tr>
<th>CALCULATED CAPACITIES FOR Memb 1 UNIT - kN,m</th>
<th>SECTION CLASS 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>MCZ= 9.2 MCY= 2.1 PC= 12.3 PT= 242.1 MB= 3.6 PV= 68.7</td>
<td></td>
</tr>
<tr>
<td>BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.132 K = 1.000</td>
<td></td>
</tr>
<tr>
<td>PZ= 242.05 FX/PZ = 0.00 MRZ= 9.2 MRY= 2.1</td>
<td></td>
</tr>
</tbody>
</table>
```
What's New?

AD.2007-08.4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

AD.2007-08.4.1 Eurocode 2:2004 Slab Design

The RC Designer module (Concrete Design mode) can now be used to design slabs per Eurocode 2 (EC2 EN 1992-1-1:2004).

- **Slab design per Eurocode 2:2004** (on page 1049) - Design of EC2 slabs is performed in essentially the same way any other slab, though you should ensure that any country-specific national annex values are properly specified.

AD.2007.08.4.2 Changes to STAAD.foundation license

The license for STAAD.foundation V8i (release 5.3) is now included with STAAD.Pro contains the capability for design the following rigid foundation types:

- Isolated
- Combined
- Pile Cap

STAAD.foundation functions as a design module similar to others included with STAAD.Pro.

Contact Bentley Systems for a full license to use all of the design capabilities for STAAD.foundation or to upgrade to STAAD Foundation Advanced V8i, which includes an improved user interface, foundation toolkit wizards, plant foundation types, and FEM analysis for mat foundations.

AD.2007-08.5 Nuclear Related Features

The following new features have been added for the NRC release.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 3) NRC (build 20.07.08.22) or higher.

AD.2007-08.5.1 Design per ASME NF 3000-2001

The design per ASME NF 3000-2001 is now available as a new design code option.

Design of members per ASME NF 3000 - 2001 requires the **STAAD Nuclear Design Codes SELECT Code Pack**.

Refer to [D1.L.4. ASME NF 3000 - 2001 & 2004 Codes](on page 1616) for additional information.

AD.2007-08.5.2 ASME NF 3000-2001 Service Level Conditions

All editions of the ASME NF 3000 code now can be designed for different service level conditions as defined by that code.
Service Level Conditions are basically the loading conditions for which the plant structure and its components are to be designed. The same primary load can be multiplied by different factors to signify the different service levels. Also the load combinations for various service levels are different and pre-defined by the code.

Each ASME NF 3000 code edition now contains the a new design parameter, SLR, which is used to specify the service condition level as defined in the codes. For the case of service level D (failure), three additional new parameters –KS, KV, and KBK– are used to directly specify the service level factors.

Design of members per ASME NF 3000 - 2001 requires the STAAD Nuclear Design Codes SELECT Code Pack. Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1625) for additional information.

AD.2007-08.5.3 TATA Structura Sections

The catalog of available TATA Structura (Indian) hot rolled steel sections has been expanded to include the hollow sections from TATA publications.

Specifying a TATA Structura hollow section

1. Select the General | Property page.
   The Properties - Whole Structure dialog box opens.
2. In the Properties - Whole Structure dialog box, click Section Database.
   The Section Profile Tables dialog box opens.
3. Select the Steel tab.
4. Select one of the tables available in the TATA Structura section: Rectangular Hollow, Square Hollow, or Circular Hollow.
5. Select an entry from the Select Profile list.
6. Click Add.

The section is now added to the Section tab in the Properties - Whole Structure dialog box and can be assigned to members.

V8i (SELECTseries 2)

The Software Release Report for STAAD.Pro V8i (SELECTseries 2) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 1) (release 20.07.06) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-07.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.
AD.2007-07.1.1 Academic Licensing

In order to ensure that the next generation of engineer that emerges from the higher education system is up to speed using our applications, Bentley has a policy of providing software to Universities and Colleges at a favorable rate.

Students can now use STAAD.Pro under an Academic License, which is obtained through a SELECT account. Contact your regional engineer or visit Bentley.com to obtain a license.

**Note:** When using an Academic License, the program window title bar and About window indicate this. Similarly, all output (Analysis files and Reports generated from STAAD.Pro) are marked as "Academic License User."

**Caution:** The Advanced Analysis Engine is not available when using the program under an Academic Licence.

AD.2007-07.1.2 StructLink and PipeLink Plug-ins

Two plug-ins are available to you when installing STAAD.Pro: StructLink and PipeLink.

- StructLink is a utility used for the bi-direction exchange of data between STAAD.Pro and ProSteel V8i.
- PipeLink is an all new utility used for the exchange of pipe stress model data between STAAD.Pro and AutoPIPE V8i. Refer to **AD.2007-07.3.4 AutoPIPE V8i (SELECTseries 2) and PipeLink support** (on page 169) for additional the bi-directional data exchange capabilities made available through this utility.

**Note:** During the installation of STAAD.Pro, select the option to install additional programs and utilities in order to have these two utilities installed.

Refer to the documentation included with these plug-ins for additional information on their use.

AD.2007-07.1.3 Structural Dashboard Integration

Bentley's Structural Dashboard is now integrated into STAAD.Pro V8i.

Bentley's Structural Dashboard V8i is a free utility application which allows you to manage workflows and project files as well as keep up to date with latest products, news, and Be Communities happenings. This program can now be accessed from within STAAD.Pro and will launch whenever STAAD.Pro is started to assist you in managing your entire project workflow.

When STAAD.Pro is first launched after installing Structural Dashboard, a welcome dialog opens to allow you to set the automatic launch option. To launch the program and continue allowing it to launch whenever a Bentley Structural program starts, leave the option selected and click the **OK** button. Otherwise, you can de-select this option before proceeding.
Welcome to Structural Dashboard

Starting the Structural Dashboard...

- Launch Structural Dashboard on program startup

Note: To turn off launching the Dashboard, disable under 'Edit Settings' in the Dashboard or the program configuration settings.

- Show me this dialog on startup

OK

Launch Structural Dashboard on program startup

Set this option to open the Structural Dashboard application whenever STAAD.Pro is opened.

Tip: This setting can be changed at any time from within the Structural Dashboard program.

Show me this dialog on startup

Set this option to display this dialog whenever STAAD.Pro is opened.

Tip: This setting can be changed at any time from the Configure Program dialog File Options tab.

To launch Structural Dashboard from STAAD.Pro

1. Select File > Structural Dashboard...

   The Bentley Structural Dashboard V8i program opens.
Note: If Structural Dashboard has not been installed, this menu item is inactive. You can download the program from http://www.bentley.com/en-US/Promo/ISM/downloads/.

Note: Refer to Section 2.3.1 of the User Interface Manual for additional help in using the Structural Dashboard with STAAD.Pro.

AD.2007-07.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

Note: Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 2) (Build 20.07.07.31).

AD.2007-07.2.1 Time History Spectrum Enhancements

New options have been added to the spectrum input for a Time History definition which allow you to output time history input data, a Response Spectrum for a Time History load, or to use frequency-spectrum pairs. These options can be added by modifying the input command file.
Two new output options are available for reporting time history input and synthetic time history ground acceleration data used by the program for a time history load with the spectrum generation option. You can control the amount of output generated (as this can be quite large) as well.

A new option has also been added to allow you to instruct the program to use frequency-spectra pairs in lieu of period-spectra pairs for the time history spectrum input.

Related Links

- TR.31.4 Definition of Time History Load (on page 2630)
- M. To generate output for time history spectrum (on page 858)
- M. To use frequency-spectra pairs in a time history load (on page 859)

AD.2007-07.2.2 Response Spectrum Signed Results and IMR Load Cases

Two methods to produce signed response spectrum results have been added to the STAAD.Pro analysis engine. The Dominant and Sign commands may be used in the input file to produce signed output. Additionally, STAAD.Pro now includes an option to automatically generate new load cases based on a specified number of modes from the response spectrum.

Signed Results

STAAD.Pro can now assign a mathematical sign (positive or negative) to the modal results by one of two means. The first method allows you to select a DOMINANT mode, the sign of which will then be applied to all other modes. The second method will produce signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign.

Individual Modal Response Case Generation

The Individual Modal Response (IMR) load cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere. You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). You can also use the Repeat Load capability to combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

The modal accelerations are multiplied by the nodal masses to produce equivalent static lateral forces for each modal load case.

Note: When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.
What's New?

STAAD.Pro V8i

Response Spectrum

Code: Custom
Combination Method: SRSS

Spectrum Table

<table>
<thead>
<tr>
<th>Period</th>
<th>Acc (m/sec²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.160603</td>
</tr>
<tr>
<td>2</td>
<td>0.17954</td>
</tr>
<tr>
<td>3</td>
<td>0.198476</td>
</tr>
<tr>
<td>4</td>
<td>0.217412</td>
</tr>
<tr>
<td>5</td>
<td>0.236348</td>
</tr>
<tr>
<td>6</td>
<td>0.255284</td>
</tr>
<tr>
<td>7</td>
<td>0.274221</td>
</tr>
<tr>
<td>8</td>
<td>0.293157</td>
</tr>
</tbody>
</table>

Spectrum Type
- Acceleration
- Displacement

Interpolation Type
- Linear
- Logarithmic

Damping Type
- Damping
- CDAMP
- MDAMP

Others
- Scale: 1
- Missing Mass
- ZPA

Signed Response Spectrum Results Options
- Dominant Mode No: 1
- Signed

Individual Modal Response Load Case Generation Options
- Generate load case(s) for first 1 mode(s) starting with Load Case no. 101

Graph

[Graph showing response spectrum data]

[Image of the Response Spectrum window]
To add Signed results to a Response Spectrum

1. Select Commands > Loading > Load Commands.
   
or
   Select the General | Load & Definition page and then click the New... button.
   
The Create New Load Items dialog opens.
2. Select the Response Spectra tab.
3. Select the Code you wish to use.

   **Note:** See below for using IMR generation options. All other parameters are same as previous versions.
4. Select the option to use the **Dominant Mode No.** to assign the same sign as the selected mode to all modes.
5. (Optional) Select to provide **Signed** results to

   **Note:** Selecting this option will not use the Dominant Mode No., but rather will create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.
6. Click the Add button to add this response spectrum load.

To add Individual Modal Response results to a Response Spectrum

1. Select Commands > Loading > Load Commands.
   
or
   Select the General | Load & Definition page and then click the New... button.
   
The Create New Load Items dialog opens.
2. Select the Response Spectra tab.
3. Select the Code you wish to use.

   **Note:** See above for options to add signed results. All other parameters are same as previous versions.

   **Note:** The Individual Modal Response case generation is not available for SNIIP II code response spectra.
4. Select the option to **Generate load cases for** ... to individual modal response load cases.
5. (Optional) Specify the number of modes for which load cases will be generated.

   **Note:** Selecting this option will not use the Dominant Mode No., but rather will create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.
6. (Optional) Specify a beginning load case number for the first primary load case generated from the IMR.
7. Click the Add button to add this response spectrum load.

Related Links

- TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688)
- Response Spectra tab (on page 3021)
- M. To add a generic response spectrum (on page 843)
AD.2007-07.2.3 Design of Class 4 (Slender) Steel Sections per S16-01

An update to the Canadian Steel Design code has been added for the design of Class 4 (slender) steel sections per CAN/CSA-S16-01. Previous versions of STAAD.Pro were capable of designing Section Classes 1, 2, or 3.

The design of slender Class 4 steel sections does not require any different actions or input. The analysis engine will determine if a section meets the criteria for a Class 4 section and then perform the necessary checks, if design checks have been requested for that member.

Methodology

Refer to D4.B.6 Member Resistances (on page 1705) for a detailed description of the methodology used in STAAD.Pro for performing the design of Class 4 sections per S16-01. A verification problem using this feature has also been added to Section 3B.10.

AD.2007-07.2.4 Von Mises Stresses per AIJ 2002 and 2005

Design per AIJ (Japanese) steel design codes has been updated to include checking members in accordance with Von Mises stress criteria in AIJ 2005. This check is a requirement for the design of steel structures in nuclear power plants in Japan.

The von Mises stress equation is calculated when the new MISES parameter has been set to a value of 1 (the default value of 0 does not check this condition). The calculated forces and moments are combined per the von Mises stress criteria.

To specify a von Mises stress check in an AIJ 2002 or AIJ 2005 design

1. Create a model with steel members.
3. Click the Define Parameters... button.
   - The Design Parameters dialog opens.
4. Select the MISES tab in the parameters list.
5. Select option 1 to instruct STAAD.Pro to perform the von Mises stress check as part of the steel design.
6. Click the Add button to add this parameter.
7. Close the Design Parameters dialog.
8. Assign the MISES parameter to members as needed, just as you would any other design parameter.

How the von Mises check results are included in the output depends on the level of detail (TRACK parameter) selected:

<table>
<thead>
<tr>
<th>Track 0 or 1</th>
<th>The von Mises stress is reported if this ratio is the critical condition.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Track 2</td>
<td>The value for ( f_m = \sqrt{\sigma_x^2 + 3\tau_{xy}^2} ) (numerator in the von Mises stress ratio equation) is displayed in the Stresses output category. When the von Mises check ratio is the critical condition, the value of the ratio is reported.</td>
</tr>
<tr>
<td>Track 4</td>
<td>Used for deflection checks only. Von Mises checks are not reported.</td>
</tr>
</tbody>
</table>
Methodology


AD.2007-07.2.5 Norsok N-004

The design of tubular steel (European round pipe sections) members per NORSOK N-004 Rev 2, October 2004 has been included in STAAD.Pro. The program will perform the member design for ultimate limit states (and optional deflection checks for serviceability). The tubular joints can also be automatically generated and checked per the code.

The NORSOK code has been added to the steel design code list available in STAAD.Pro. Selecting this code allows you to assign parameters, including defining the water level above the origin (for calculating hydrostatic pressure) or the

Note: N-004 refers to the superseded version of Eurocode 3 (DD ENV 1993-1-1) in several places. In such cases, the corresponding clause from the latest version of EC-3 (EN 1993-1-1:2005) has been used in the STAAD.Pro implementation.

To perform a member design per the NORSOK N-004 code

1. Create a model with steel tubular members.
   
   Caution: The Norsok code only supports pipe sections. Errors will be presented in sections other than pipe members are used.

3. Click the Define Parameters button.
4. The Design Parameters dialog opens.
5. Specify parameters as required.
   
   Note: The height of water level above the origin is specified using the HYD parameter. Alternatively, the PSD parameter may be used to define the water pressure.

7. Assign the torsion-related parameters to members as needed, just as you would any other design parameter.

To perform a joint check per the NORSOK N-004 code

1. Add the CHECK JOINT command to a new PARAMETER manually in the STAAD.Pro input file using the Editor.
2. Perform a preliminary design by selecting Analyze > Run Analysis....
   
   The program creates an external text file titled filename_JOINTS.NGo which contains the automatically generated chord and brace definitions associated with the nodes included in the CHECK JOINT command. All joints are classified as Y by default.
3. Open the text file using a text editor program (i.e., Notepad or STAAD Editor).
4. Manually edit the joint classifications as needed.
5. (Optional) Edit the Brace and Chord definitions as needed.
**Note:** The Brace and Chord members at each joint are assumed based on the relative cross section dimensions. Lengths of Chord and Brace members are taken as the analytical beam member length.

6. Save the text file and the re-analyze the structure

**Methodology**

Refer to D1.2B. Norwegian Codes - Steel Design per NORSOK N-004 (on page 2095) for a detailed description of the methodology used in STAAD.Pro for performing steel tube member design per NORSOK N-004.

**AD.2007-07.2.6 EC3 Torsion Design**

Design per EC3 [EN 1993-1-1:2005] has been enhanced to include the design of members subject to torsion. You may select to have the program execute basic or detailed torsion stress checks. Torsion design checks can be performed on I-sections, H-Sections, Channel sections, and structural hollow sections (RHS, SHS, CHS).

**Note:** The default behavior is to neglect torsion. The new TORSION parameter must be set to either 1 (basic) or 2 (detailed) to perform torsion design.

**To include torsion design for EC3 steel design members**

1. Create a model with steel members.
2. Select **EN 1993-1-1:2005** for the **Current Code** on the **Design | Steel** page.
3. Click the **Define Parameters...** button.
   - The Design Parameters dialog opens.
4. Select the **TOR(sion)** tab in the parameters list.
5. Select either option 1 (von Mises check excluding warping effects) or option 2 (detailed checks including warping effects) to include design for torsion and click the **Add** button to add this parameter.
6. Specify the loading and support conditions of members subject to torsion using the **CMT** tab in the parameters list and click the **Add** button to add this parameter.
7. (Optional) For the cases of a concentrated torque (CMT = 2, 3, or 6) somewhere along the member length (other than the default of mid-span), specify the location of the torque using the **ALH** tab in the parameters list and click the **Add** button to add this parameter.
8. (Optional) Specify the effective length of members for torsion using the **EFT** tab in the parameters list and click the **Add** button to add this parameter.
10. Assign the torsion-related parameters to members as needed, just as you would any other design parameter.

Torsion design in EC3 is given in Cl. 6.2.7 of EN 1993-1-1:2005. Therefore, this clause is used primarily for this implementation.

EN 1993-1-1:2005 does not deal with members subject to the combined effects of torsion and lateral torsional buckling. However, EN 1993-1-6 considers such a condition in Appendix A. Therefore, STAAD.Pro uses Appendix A of EN 1993-1-6 to check for members subject to combined torsion and LTB.

The following clauses from EC3 are then considered:

- Cl 6.2.7(1)
- Cl 6.2.7(9)
- Cl 6.2.7(5)
- EC-3 -6 App A
When torsion design is included (TOR = 1 or 2), then the EC3 design output includes the following sections:

- **Basic (TORSION = 1)** - The ratio calculated for stress interaction per EC-6.2.7(5) is displayed for each load case, along with the calculated values of axial force, shear in Y and Z, Bending about Y and Z, and torsion.
- **Detailed (TORSION = 2)** - The additional clauses viz. 6.2.7(1), 6.2.7(9) and EC3-6 A-1 will be included in the output. The stress interaction ratio per each is displayed for each load case, along with the calculated force and moment values used. Additional torsion calculation details are provided as well.

**Note:** If Torsion design is selected for a member which does not have any torsional moment, a warning is displayed in the output.

**Methodology**

Refer to [D5.C.5.4 Members Subject to Torsion](on page 1762) for a detailed description of the methodology used in STAAD.Pro for performing torsion stress checks per EC3.

**AD.2007-07.2.7 Eurocode 3 National Annex**

Two additional country’s National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Finland and Poland. As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.

To Initiate a EC3-Finnish NA Steel Design, use the following procedure.


1. In the Modeling mode, click the **Design > Steel** tab.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters...** button. The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(5) Finland**.
6. Click the **Add** button.

This will insert the following commands into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 5
```

**Note:** For additional information, please refer to **D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005]** (on page 1754) and **D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005]** (on page 1792)

When the Finnish National Annex to EC3 is used for design, the output section title is revised to include the Finnish National Annex (National Annex to SFS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Finnish NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

AD.2007-07.2.7.2 Polish National Annex to Eurocode 3 (EN 1993-1-1:2005)

To Initiate a EC3-Polish NA Steel Design, use the following procedure.

The Polish National Annex document referred to is “National Annex to Standard PN-EN 1993-1-1”.

1. In the Modeling mode, click the **Design > Steel** tab.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters...** button. The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(6) Poland**.
6. Select the new **PLG** parameter in the list box.

```
Note: This parameter is used to select if additional checks per clause 6.3.3 will be performed for designs using the Polish National Annex.
```

7. Click the **Add** button.

This will insert the following commands into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 6
```

**Note:** For additional information, please refer to **D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005]** (on page 1754) and **D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005]** (on page 1792)
When the Polish National Annex to EC3 is used for design, the output section title is revised to include the Polish National Annex (National Annex to PN-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Polish NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

**AD.2007-07.2.8 AS4100 Physical Member Design**

The workflow for design steel members per AS 4100:1998 has been updated to incorporate the use of physical members. Physical members are groups composed of a series of analytical beam elements of the same section and which are colinear (analytical beams are the beams used in modeling in STAAD.Pro). Using physical beams allows you to design for the actual conditions of the structure and assign specifications based on the true conditions of a steel member.

Some of the physical member design updates apply to design codes other than AS 4100, such as checks in the STAAD.Pro analysis engine for physical member overlapping and colinearity. These checks were previously made in the graphical interface but now they will checked again in the engine in the event you have manually generated the STAAD.Pro input file.

The physical member mode is initiated through the Toggle Physical Member mode tool found in the Steel Design toolbar (which docked on the left hand side of the screen by default). This “mode” is used when modeling the structure and any parameter or specification added while this tool is toggled on will then only be available for physical member groups.

**Tip:** Command entries in the tree, material properties, and specifications will be designated with “(Physical)” when added in this mode.

Physical members are then formed or selected using the tools in the Steel Design toolbar. Refer to [M. Physical Members](on page 649) for additional information.

**Physical Member Restraints**

A new parameter has been added for AS 4100 physical members to describe the bracing conditions/locations on a physical member. This parameter describes where the restraint is located along the length of the physical member and the type of restraint on the top or bottom flange.
Engine Physical Member Validation

When creating physical members, the STAAD.Pro graphical interface will check to ensure that the analytical members included in a physical member definition are both interconnected and colinear. However, it is not uncommon for input files to be generated outside of the STAAD.Pro graphical environment. Thus, these checks are now performed by the STAAD.Pro Analysis & Design engine again when an analysis is performed. You will be alerted if either condition is not met.

**Tip:** The analytical members contained in a physical member definition must be colinear (or, all lying in a straight line). Each adjacent analytical member must be within 5° of one another to meet this condition.

Refer to D2.B.8 Design Parameters (on page 1641) for additional information using the new SGR (steel grade) and LHT (load height position) design parameters for steel design per AS4100. Refer to D2.B.12 Physical Member Design (on page 1649) for additional information on using the PBRACE parameter and performing the design of physical members per AS4100.

**AD.2007-07.2.9 SNiP 2.23-81 Steel Design**

Several minor enhancements have been made to STAAD.Pro regarding steel design per the SNiP 2.23-81 code.
The following corrections and enhancements were made to the SNiP 2.23-81 steel design code implementation in STAAD.Pro:

- PHI and NIU factors messages extensions explaining different design results cases
- Additional bending check for non axial compression/tension with small eccentricity
- Compressed steel member design by weakened section
- Full Check by combinations envelope for different sections of each steel member. Print of Analysis results for each member section
- Additional parameters for Steel grade by EC3 in EN 10025-2 steel tables
- Correction of other minor bugs and errors

AD.2007-07.2.10 Geometric Nonlinear Analysis Cycle Control

You are now able to limit the analysis cycle using a displacement limit control. These controls can be found on the Analysis/Print Commands dialog Nonlinear Analysis tab or input manually in the input command file.

The displacement limit control allows you to select a nodal displacement degree of freedom be monitored during a geometric nonlinear analysis. A target displacement is set and, if the number of load steps set is two or greater, the analysis will proceed step-by-step until the target displacement is met or exceeded. This provides you with an additional, practical means of limiting the number of steps used in a geometric nonlinear (GNL) analysis.

Related Links
- TR.37.8 Geometric Nonlinear Analysis (on page 2830)
- Nonlinear Analysis tab (on page 3071)
- A. To specify a nonlinear analysis (on page 930)

AD.2007-07.2.11 Jindal Steel Section Database

To add a section from the available JPSL catalog, use the following procedure.

A number of Jindal Power & Steel Limited (JPSL) catalog sections have been added to the section database.

1. Select the General | Property page.
2. In the Properties - Whole Structure dialog, click Section Database. The Section Profile Tables dialog opens.
3. On the Steel tab, select the Jindal entry in the table families list.
4. Select the table, section, and type specification.
5. Click Add.

AD.2007-07.2.12* Design per ASME NF 3000 2004 Code

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented per the 2004 edition of this code and the Design | Steel page has been updated to allow the design parameters to be defined and assigned.

To perform a steel design per ASME NF 3000 2004

Use the following procedure to specify post analysis steel design code checking requirements for the ASME NF code.

1. Create a model with steel members.
2. Select **ASME NF3000 2004** for the **Current Code** on the **Design | Steel** page.

3. Click **Define Parameters**...

The **Design Parameters** dialog opens.

4. Specify parameters as required.

5. Close the **Design Parameters** dialog.

6. Assign parameters to members as needed.

7. Select Analysis > Run Analyze (or press CTRL+F5).

For more information on the technical requirements of this design code, including the full set of parameters and default values, refer to **D1.L.4. ASME NF 3000 - 2001 & 2004 Codes** (on page 1616).

**Note:** The STAAD Nuclear Code pack is required to perform designs per an ASME NF 3000 code.

---

**AD.2007-07.2.13* Update to ANSI AISC N690 1984 & 1994 Codes**

Four new Stress Limit Coefficients (SLC) parameters have been added for designs per ANSI/AISC N690 1984/1994 codes.

These parameters, SFC, SFT, SMZ, and SMY, all default to 1.0 and are used to control the interaction equations in Section Q1.6 of the ANSI/AISC N690 1984/1994 codes.

Equations Q1.6-1a, Q1.6-1b, Q1.6-2 and Q1.6-3 of code ANSI/AISC N690 1994 will be rewritten as follows:

- Members subjected to both axial compression and bending stresses are proportioned to satisfy equation Q1.6-1a:

  \[
  \frac{f_a}{F_a} + C_{my} \frac{f_{by}}{F_{by}} + \left[C_{mx} \frac{f_{bz}}{F_{bz}} + \left[(1 - \frac{f_a}{F'_e}) \frac{F_{by}}{F_{bz}}\right]\right] \leq 1.0
  \]

  and Q1.6-1b

  \[
  \frac{f_a}{(0.6 \cdot F_y)} + \frac{f_{by}}{F_{by}} + \frac{f_{bz}}{F_{bz}} \leq 1.0
  \]

  when, \( f_a / F_a > 0.15 \), as per section Q1.6.1 of the code.

- Otherwise, equation Q1.6-2 must be satisfied:

  \[
  \frac{f_a}{F_a} + \frac{f_{by}}{F_{by}} + \frac{f_{bz}}{F_{bz}} \leq 1.0
  \]

- Members subjected to both axial tension and bending stress are proportioned to satisfy equation Q 1.6-1b:

  \[
  \frac{f_a}{(0.6 \cdot F_y)} + \frac{f_{by}}{F_{by}} + \frac{f_{bz}}{F_{bz}} \leq 1.0
  \]

Refer to **D1.K.2.1 Design Process** (on page 1582) for additional information on using the ANSI N690 1984 and 1994 codes.

---

**AD.2007-07.2.14* Load Combination Enhancements**

It is now possible to refer to a previously defined load combination within a new load combination. For example, a SRSS combination of individual response spectrum cases can now be referenced in a load combination along with dead load, live load, etc.

There are no changes to the input file syntax. Load combination definitions may now refer to existing load combination numbers along with load case numbers.

**Note:** There is no limit to the amount of load combination “nesting” which can be done in STAAD.Pro, other than the total limit of load cases and load combinations allowed by the program.

---

**Related Links**
**What's New?**

**STAAD.Pro V8i**

- **TR.35 Load Combination Specification** (on page 2791)
- **Define Load Combinations dialog** (on page 3003)
- **M. To define a new load combination** (on page 866)

**AD.2007-07.2.15* Enhancement to Maximum Number of Response Spectrum Load Cases**

STAAD.Pro now supports up to 50 response spectrum load cases, instead of the previous limit of four. Refer to **TR.32.10.1 Response Spectrum Analysis** (on page 2687) for additional information on using Response Spectra.

**AD.2007-07.3 Features Affecting Post Processing**

Several new features have been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

**AD.2007-07.3.1 RAM Connection V8i (SELECTseries 1) Support**

The enhancements included in Bentley’s RAM Connection V8i (releases 6 and 7) are now available in STAAD.Pro. This includes new connection types, new codes, and design for seismic loads.

Some of the new features and enhancements include:

- Now compatible with versions of RAM Connection V8i up through release 7.0 (SELECTseries 3).
- British Design Code - Connection design per BS5950-1:2000 (British standard) has been added. This code can now be selected along side AISC codes.
- AISC Seismic Provisions - The seismic provisions of AISC 341-05 have now been added for connections per AISC codes.
- Seismic Frame Management - A new Seismic Frames page has been added to assign the lateral seismic resisting system classification to frames for connection design. This page is also used to add plastic hinge locations to beam members.
- Base Plate Design is now available for AISC (ASD & LRFD) connections. Column base plates are available in the Smart Connections dialog and Column-Brace gusset base plates are available in the Gusset Connections dialog.
- The RAM Material dialog has been expanded to accommodate for various materials from different countries. UK steel, bolt, and weld types have been added, as well as concrete and anchor bolt materials for US base plate design.

**Note:** A notification message may be displayed when selecting the RAM Connection mode that you need to provide some additional material properties.

- Several new selection methods have been added to the **Select > By Joints** > sub-menu.
- Reports have been enhanced with code references and display of formulas used.

**AD.2007-07.3.2 RC Designer**

Several new features have been added and existing features have been modified in the RC Designer mode. These are explained in the following pages.

- **ACI 318M-05 (Metric)** (on page 1218) - Uses “soft” conversions for US standard bar sizes. Therefore, bar sizes are unchanged from US Customary units, but named for the nearest metric equivalent. The conversion for material properties is performed similarly.
**Note:** Mixed units cannot be used for designs per ACI 318-05. The metric version is listed as separate design code when creating design briefs.

- **GB50010** (on page 1318) - Concrete beam and column design per GB50010 is now available.
- **IS456** (on page 1325) has been updated to the 2000 edition and now includes an option to perform additional reinforcement detailing checks for seismic conditions per **IS 13920:1993** (on page 1061). This code is used for the ductile detailing of reinforced concrete structures subjected to seismic forces.
- Documentation has been expanded for the **SP52-101-03 design briefs** (on page 1342) and **technical reference** (on page 1066).

**AD.2007-07.3.2.1 ACI 318 Metric**

A new design code has been added for beams, columns, and slab element designs in the RC Design module for the metric version of ACI 318-05.

ACI 318M-05 is the standard published by the American Concrete Institute which describes the equations to be used for metric design. It differs from the normal ACI 318-05 standard in that it has been converted to a “soft metric” form, where conversions have been applied to the numbers used in the formulae and to the bar sizes, and then some rounding has been done. This means that the bars to be used are the same as with US customary units, but they are quoted to the nearest millimeter.

**Note:** Refer to **D. American Concrete design per ACI 318M - 2005** (on page 1105) for additional information.

**Note:** A valid license for the U.S. Design Codes (Standard) package is required to use this feature.

To create a ACI 318M-05 Design Brief for columns or beams

1. Open a model in RC Designer.
2. Select the Groups/Briefs Page in the page control.
3. Click the **New Brief** button at the bottom of the Design Briefs table.
   - The New Design Brief dialog opens.
4. Enter a title in the **B#** field.
5. Select **ACI 318M-05** in the **Design Code** list.
6. Select the appropriate **Design Type** (either Beam or Column).
7. Click the **OK** button.
   - The corresponding ACI 318M-05 Beam Brief or ACI 318M-05 Column Brief dialog opens, depending on the choice made above.
8. Specify parameters as needed.

   **Note:** The design briefs are identical to those for ACI 318-05 checks with all entries being input in “soft” metric units as indicated.

9. Click the **OK** button.

**AD.2007-07.3.2.2 GB50010**

Concrete beam and column design per the GB 50010-2002 code is now available.

The GB50010 checks are initiated by selecting this code for the design brief and selecting the appropriate parameters. Refer to **D. Using GB50010 (Chinese) Concrete Member Design** (on page 1318) and **D. Chinese Concrete design per GB50010** (on page 1025) for additional information on using this new feature.
**Note:** A valid license for the Asian Design Codes package is required to use this feature.

To create a GB50010 Design Brief for columns or beams

1. Open a model in RC Designer.
2. Select the Groups/Briefs Page in the page control.
3. Click the **New Brief** button at the bottom of the Design Briefs table.
   - The New Design Brief dialog opens.
4. Enter a title in the **B#** field.
5. Select **GB50010** in the **Design Code** list.
6. Select the appropriate **Design Type** (either Beam or Column).
7. Click the **OK** button.
   - The corresponding GB50010 Beam Brief or GB50010 Column Brief dialog opens, depending on the choice made above.
8. Specify parameters as needed.
9. Click the **OK** button.

---

**AD.2007-07.3.2.3 IS456 with Seismic Design per IS13920**

Design of columns and beams per IS 456 is now performed per the 2000 edition of that code. The option to perform additional reinforcement detailing checks for seismic conditions per IS 13920:1993 is also now available. This code is used for the ductile detailing of reinforced concrete structures subjected to seismic forces. The IS456 checks are initiated by selecting this code for the design brief and selecting the appropriate parameters. Additional seismic checks per IS 13920 are specified by selecting this option on the design brief General tab. Refer to D. IS 13920 Seismic Design (on page 1061) and D. IS456 (on page 1325) for additional information on using this new feature.

**Note:** A valid license for the Indian Design Codes package is required to use this feature.

To create an IS456 Design Brief for columns or beams

1. Opens a model in RC Designer.
2. Select the Groups/Briefs Page in the page control.
3. Click the **New Brief** button at the bottom of the Design Briefs table.
   - The **New Design Brief** dialog opens.
4. Enter a title in the **B#** field.
5. Select **IS456** in the **Design Code** list.
6. Select the appropriate **Design Type** (either Beam or Column).
7. Click the **OK** button.
   - The corresponding IS456 Beam Brief or IS456 Column Brief dialog opens, depending on the choice made above.
8. Specify parameters as needed.
9. Select the option to **Check for IS 13920** on the General tab to perform additional seismic design.
   - **Note:** See the following procedure for designing for IS 13920.
10. Click the **OK** button.
    - Design longitudinal and shear reinforcement in beams per IS13920
1. In the **IS456 Beam Design Brief** dialog General tab, select the **Check for IS13920** option.

2. Perform a beam design.

3. Select the **Earthquake** page.

4. Select the appropriate load case is selected to perform the seismic checks to allow for a strong column-weak beam mechanism.

   The program internally calculates the moment capacities of the beams and columns framing into a joint and assign a "pass/fail" status depending on the corresponding beam and column capacities at that joint for the direction under consideration.

5. Select the **Earthquake | Reinforcement** page to review the status of the IS 13920 design.

**AD.2007-07.3.3 Enhanced Geometric Nonlinear Post Processing**

The graphical display of the results of a geometric nonlinear analysis have been improved. The Graphs are now easier to read and a new control has been added to limit the maximum Load Step displayed.

When a structure is analyzed with a geometric nonlinear analysis, the nodal displacements can be viewed by selecting the Node | Nonlinearity page in the Post-Processing mode. The Node Displacements Curve has been enhanced and a new control has been added to the Node Displacements table to limit the maximum Load Step plotted.

In the event that a load level is specified which exceeds the non-linear buckling capacity of a structure, the analysis performed by STAAD.Pro will produce exceedingly large post-buckling displacements. This signifies that the load steps in this post-buckling level are beyond the scope of designed performance of the STAAD engine (and likely beyond the load level and effects intended by the engineer). In these cases, the scale of the non-linear load displacement curve was did not adequately display the pre-buckling characteristics due to the very large scale required to display the post-buckling displacements.

Now, in the event of a large post-buckling load, the maximum load level scale of the Load Level vs. Displacement graph can be limited to the currently selected load step by selecting this option on the Node Displacements table, thus allowing you to see the pre-buckling behavior.
AD.2007-07.3.4 AutoPIPE V8i (SELECTseries 2) and PipeLink support

STAAD.Pro is now capable of two-way data exchange with AutoPIPE via a new PipeLink utility. Additionally, the tools for transferring loads between the pipe model and the structural model have been enhanced such that you can now select, update, and remove loads to be applied to the structural model.

**Note:** For details on the updates to Bentley AutoPIPE V8i, refer to the documentation included with that product.

Some of the new features and enhancements include:

- The integration between STAAD.Pro and AutoPIPE has been enhanced to allow bi-directional transfer of data between STAAD.Pro and AutoPIPE. This is accomplished through a new plug-in program called PipeLink, which is used to generate common database files for use in both AutoPIPE and STAAD.Pro. [Exporting data](on page 3110) back to AutoPIPE is now possible.
- The system now allows pipe models to be imported from a number of databases and these models may all be stored locally. The active model can be selected from the [Pipe Model dialog](on page 3111).
- The loading transfer process has been significantly revised. The mappings between the pipe loadings and the generated structure loadings (on page 3119) are recorded so that these structure loadings may be revised or removed rather than the previous once-only method. Further, loads can now be modified individually.
- Connections can now be individually selected to transfer moment to the supporting STAAD model structure by selecting the [Lever Arm](on page 3117) option at each connected Pipe Node.
- The program now connects pipe supports – rather than pipe nodes – to the structure. This allows both node and support labels to be used. This allows additional filters in the [connection wizard](on page 3113) and also allows STAAD.Pro to import multiple supports at a single support node.
Pipe supports are now graphically displayed with icons which reflect the nature of the pipe support. The display of these icons, along with other pipe model elements, can be controlled through a new tab in the Diagnose dialog (on page 3121).

**AD.2007-07.3.5 Transverse IRC Loading in STAAD.Beava**

For the specific bridge width, IRC (Indian Road Congress) chapter 6-2000, table 2, clause 207.4 defines the rules to combine the live loads. This new feature allows you to either use these IRC live load rules or use an iterative, custom method. If the IRC rule option is selected, this function uses the appropriate live loads and number of design lanes and the generates all the possible load combinations as stated for this particular bridge width. Otherwise, you can select a specific live load and design lane to generate combinations.

Loading rules per IRC Chapter 6 are applied in much the same way as previous codes. The defined roadway for the selected deck(s) is divided into design lanes and the selected load class is applied to the structure achieve the specified actions.

*To specify loading per IRC Chapter 6*

1. Open an analyzed bridge model in the Bridge Deck mode.
2. Create Deck and Roadway definitions.
3. Generate influence surfaces for the structure.
4. Select **Loading > Run Load Generator**.

   The Load Generation Parameters dialog General tab opens.

5. Select IRC Chapter 3 for the Design Code and select the appropriate Limit State.

   The <code> tab updates to display IRC Loading.

6. Select the **IRC Loading** tab.

   ![Image of IRC Loading tab]

7. Select the appropriate **Loading Class**.

   **Note:** Combinations of the AA, B, and 70R vehicles have been added to the included vehicle definitions. These may be reviewed in the Vehicle Database dialog.

8. (Optional) Specify an impact factor or modify the **Multiple Presence Factors** as needed.
9. Specify the decks for consideration on the Decks tab and the load effects to dictate load placement on the Node Displacements, Support Reactions, Plate Center Stresses, and Beam End Forces tabs.

10. Click the OK button.

The program places the selected loads in design lanes to produce the maximum or minimum effects requested. A text file containing a summary of the generated loads and corresponding effects is opened in a text editor for review.

Related Links
- IRC Specific Parameters (on page 3135)

AD.2007-07.3.6 STAAD.foundation V8i Integration

The export of support geometry and reactions to STAAD.foundation V8i can now be initiated from within STAAD.Pro using the Foundation Design mode. This feature is similar to the Import STAAD.Pro File capability included in STAAD.foundation.

When selected, the Foundation Mode opens the Foundation page which contains a view of the whole structure and the Foundation Design Options dialog.

From here, you can select to include all supports, you can graphically select supports, or you can specify a list of support numbers for exporting to a STAAD.foundation project. Similarly, the load cases from the analysis are listed for inclusion in the STAAD.foundation project.

Tip: Models containing a large number of supported nodes or load cases may result in slow performance on older computer hardware. Exporting a limited set of data can be used to improve performance in STAAD.foundation in these cases.

Planned future enhancements also include the export of mat foundations modeled in STAAD.Pro for design in STAAD.foundation.

AD.2007-07.3.7 Additional Section Databases in RAM Connection mode

Steel section databases for the following countries are now available for use when design connection in RAM Connection mode:

- Indian
- European
- Japanese
- Australian

Note: Connection design is only performed per the US and British codes available in RAM Connection.

AD.2007-07.4 Additional Features

The following features have yet to undergo testing and are presented "as is."

Note: Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 2) (Build 20.07.07.32).

Beta Features

The following features have yet to undergo testing and are presented "as is."
AD.2007-07.4.1* Design of Class 4 “Slender” Sections in IS800:2007

The design of slender classified sections (only rolled or welded I sections) per IS:800-2007 has been added to STAAD.Pro.

The IS:800-2007 code does not provide any clear guidelines about what method should be adopted for the design of slender section. The “Flange Only” concept has been adopted where it is assumed that flexure is taken by the flanges alone and the web will resist shear with adequate shear buckling resistance. This means that the flange elements must be non-slender with slender web element to qualify for slender section that can be designed. If any of the flanges become slender, the design will not be performed for Bending and a warning message is displayed.

Refer to D8.E. Indian Codes - Steel Design per IS 800 - 2007 (on page 1942) for additional information on the design procedures used for slender sections for IS800:2007 as well as a verification example problem.

V8i (SELECTseries 1)

The Software Release Report for STAAD.Pro contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (release 20.07.05) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-06.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

AD.2007-06.1.1 CIS/2 Translator Update

The STAAD.Pro tool to import and export models with the CIS/2 translator has been enhanced for the transfer of models into 3D modeling, such as Intergraph SmartPlant® 3D (SP3D).

The CIS/2 (CimSteel Integration Standard, Version 2) allows for the transfer of steel models using a prescribed data standard in the STEP (Part 21) format. These files can contain different models including analysis models. In previous versions of STAAD.Pro, CIS/2 files could not be imported into an existing STAAD file. The import process has now been updated so that new STAAD input files can be created or existing input files updated from a CIS/2 file.

STAAD.Pro will retain all relevant information generated by SP3D – including the object IDs (GUIDs) – when importing CIS/2 files. Further modeling operation will be done in STAAD.Pro which includes special purpose load generations, analysis, design and member selections and modifications. You may then export out to a CIS/2...
STEP file and retain all information inherited from SP3D STEP file and addition/deletion/modification information performed in STAAD.Pro.

Using import and export of STEP files in SP3D, further modification can be made in SP3D and the STAAD.Pro model can be updated its model. Only geometry, member properties, boundary condition information are within the update scope of STAAD.Pro. While updating the STAAD.Pro model, no other information will be considered. This round-trip process can be repeated an unlimited number of times.

Additionally, two new VBS Macros have been included in the User Tools to aide in the verification of model integrity before and after update operations using the Import dialog. These may be found in the User Tools drop-down menu in the File toolbar or under Tools > User Tools.

**New User Tools (VBS Macros)**

Two VBS macro are developed to assist users to confirm the model integrity before and after the update operations.

- **List Object GUIDs**
  
  Opens the GUID Report tables. This is useful to verify that the GUIDs are same as those in SP3D.

- **CIS/2 Object Update Report Tool**
  
  Used to capture model state before an update operation (Create Pre-Update Report) and then same tool can be invoked again after the update process to generate a report showing what exactly has been updated (Create Post-Update Report).

**Related Links**

- [I. To import a CIS/2 file](#)
- [I. To export to a CIS/2 file](#)
AD.2007-06.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2007-06.2.1 ANSI/AISC N690-1984 Design Code

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the ANSI/AISC N690-1984: Nuclear Facilities - Steel Safety-Related Structures for Design, Fabrication, and Erection.

The parameter CODE AISC N690 1984 is used to initiate code checking per ANSI/AISC N690-1984.

The full details for this code - including parameters, commands, and technical background - are in D1.K.2. ANSI/AISC N690-1984 Code (on page 1582).

To use the ANSI/AISC N690 1984 code

1. In the modeling mode, select the Design | Steel page.
2. Select AISC N690 1984 in the Current Code drop-down list.

AD.2007-06.2.2 Update to Russian Concrete Design

The Russian SNiP concrete design routines have been updated to accommodate new reinforcement and concrete class definitions. In order that these new classes can be assigned to members that are to be designed, the following changes have taken place in the RCL, BCL, and RHS parameters.

Refer to D13.A. Russian Codes - Concrete Design Per SNiP 2.03.01-84* (on page 2113) for additional information.

Additionally a few other minor updates have been incorporated to ensure axial tension is ignored in column design and that the provided area of steel in both directions is not less than the minimum.

Note: STAAD.Pro supports design of concrete beams and columns to Russian SP52 code is supported in the interactive RC Designer module (Concrete Design Mode in the GUI). For more details of the RC Designer see D. Interactive Concrete Design (on page 1005). For details on the SP52 details see "AD.2007-1001.4.1 Beam and Column Designs to the Russian Concrete Code SP52."

To select reinforcement or concrete class definitions

1. In the modeling mode, select the Design | Concrete page.
2. Select **SNiP 2.03 01-84** in the **Current Code** drop-down list.
3. Click the **Define Parameters** button below the model outline panel.
4. The **RCL**, **BCL**, and **RSH** parameters have the new definitions available. Refer to [D13.A.2 Design Parameters](on page 2114) for additional information.

   Reinforcement Class for longitudinal reinforcement (RCL):
Compression class of concrete B (BCL):

Reinforcement class for shear reinforcement (RSH):
AD.2007-06.2.3 Eurocode 3 National Annex

Two country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: United Kingdom and France. As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.
Caution: The GB1 parameter (which, in fact was common to the base EC-3 and was a reminiscent of the previous DD ENV implementation of EC-3) has been removed. Hence any legacy STAAD files that have the GB1 Parameter defined will need to be revised to take out this parameter as it is no longer valid as per the latest EN1993.

AD.2007-06.2.3.1 United Kingdom National Annex to Eurocode 3 (EN 1993-1-1:2005)

To Initiate a EC3-UK NA Steel Design, use the following procedure.

The UK National Annex document referred to is "NA to BS EN 1993-1-1:2005".

1. In the Modeling mode, click the Design | Steel page.
3. Click the Define Parameters button.
   The Design Parameters dialog opens.
4. Select the NA parameter in the list box.
5. Select the option for (1) United Kingdom.
6. Click Add.

This will insert the following command into the STAAD input file:

CODE EN 1993-1-1:2005
NA 1

Note: For additional information, please refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754) and D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1792)

AD.2007-06.2.3.2 French National Annex to Eurocode 3 (EN 1993-1-1:2005)

To Initiate a EC3-French NA Steel Design, use the following procedure.

The French National Annex document referred to is "Annexe Nationale a la NF EN 1993-1-1:2005".

1. In the Modeling mode, click the Design | Steel page.
3. Click the Define Parameters button.
   The Design Parameters dialog opens.
4. Select the NA parameter in the list box.
5. Select the option for (4) France.
6. Click Add.

This will insert the following command into the STAAD input file:

CODE EN 1993-1-1:2005
NA 4

Note: For additional information, please refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754) and D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1792)
AD.2007-06.2.4 Chinese Static Seismic Loading

A simplified base shear method of the seismic load generation for the Chinese GB50011-2001 code has been added to STAAD.Pro V8i.

This set of commands may be used to define and generate static equivalent seismic loads as per Chinese specifications GB50011-2001. This load uses a static equivalent approach, similar to that found in the UBC. Depending on this definition, equivalent lateral loads will be generated in the horizontal direction(s).

Related Links
- TR.31.2.5 Chinese Static Seismic per GB50011-2001 (on page 2563)
- Add New Seismic Definitions dialog (on page 3044)
- M. To add a seismic load definition (on page 840)

AD.2007-06.3 Features Affecting the RAM Connection Design Mode

Several new features have been added and existing features have been modified in the RAM Connection Design Mode. These are explained in the following pages.

Note: Full use of the RAM Connection Mode requires access to a valid RAM Connection license. If you do not possess a license, contact your Bentley account manager to have it added to your SELECT licenses. Without a valid license, only a small subset of the full range of available RAM connections can be utilized.

AD.2007-06.3.1 RAM Connection V8i Support

The enhancements included in Bentley’s RAM Connection V8i (release 5.5) are now available in STAAD.Pro. Additionally, the connection assignment and design process has been streamlined within STAAD.Pro. Now, joints and connections can be automatically assigned for a set of selected members and connection designs can be grouped together. Additionally, when a connection is edited using the RAM Connection pad, those changes will be saved in the STAAD.Pro model design.

Connections are designed in the newly updated RAM Connection Mode by creating “Joints,” from the geometry, section properties and forces resulting from the analysis and assigning a design brief made up of connection templates. A suitable connection design, if one is available, will be reported once you have selected the appropriate connection templates.

In previous releases of STAAD.Pro, you were able to only select and assign a connection to individual joints. Now, any number of joints may be selected and designed. Further, connection design is performed automatically for you once appropriate templates have been selected for the selected joints. These enhancements greatly reduce the time required for connection design in models of all sizes.

Tip: The selected load envelope is now used for all connection designs, instead of per design brief as in previous versions of STAAD.Pro.
AD.2007-06.4 Features Affecting the Piping Mode

STAAD.Pro can utilize the pipe layout and reactions created in the applications ADLPipe or AutoPipe. The pipe model can be imported in the Piping Mode. The following describes the method of using the Piping Mode and features recently added to this mode.

AD.2007-06.4.1 AutoPIPE Integration Enhancements

An enhancement to how pipe/structure connections are assigned has been made to the Piping module. A new Support Connection Wizard is available to allow you to add multiple supports to the entire model or a subset of a model, based on some general parameters.

Note: Pipe-structure links are not part of the undo system. Nodes created at the end of the wizard will be removed by an undo but the links are not changed.

Within the Piping mode, after the pipe model has been loaded, you will be able to call up a modeless connection wizard. The wizard will take you through the following steps:

1. Defining the set of pipe nodes to consider.
2. Defining the set of structural beams to consider.
3. Defining the set of structural nodes to consider.
4. Setting range and tolerance parameters.
5. Previewing and accepting the determined connections.

Potential connections will be determined after step 4 and fully created at the end of step 5.

Potential connections will be determined by finding the closest beams and closest nodes to each pipe node. In the previewing stage the closest five items, along with a “to ground” option, will be available as options to you, sorted by distance.

Parameter Page / Connection Finder

The connection finding routine runs in two parts. First looking at beams (length of perpendicular from beam) and then looking at nodes (straight line length). The five closest found connectable points are saved to be presented on the results page. In order to provide control over the connection finder, several parameters are available to you for editing. The default values of distance-based parameters will depend on the base unit of STAAD.Pro.

General

- Max. Range: double: default 2m / 6’ : Potential connections beyond Max. Range will be discarded.
- Insert nodes into beams at connection points: default true : This parameter effects the connection of the structures rather the point finding algorithm. If set true a new node will be created at each intermediate beam point and the connection made to that rather than to the beam itself. If the algorithm finds new node points within “End Tolerance” of each other then only one new node will be added.

Beam

- End Tolerance : double: default 5cm /2” : To allow for differences in precision and to avoid very short beam breaks this parameter will determine at what distance from the node the perpendicular will be considered to be at the node itself.
Allow Non-Perpendicular Connection at End Nodes: Boolean : default true: This is only really relevant if the node subset does not explicitly include the nodes at the end of members in the beam subset. If set "true" the beam end nodes will be included in the node search. If set 'false' the end nodes of a given beam will only be considered for connections perpendicular to the beam, unless they have been explicitly added to the node subset.

This page has no effect on structure diagrams. The connection finding routine is run when advancing from this page.

Related Links
- M. To use the Support Connection Wizard (on page 879)
- Support Connection Wizard (on page 3113)
- I. To export structure date to AutoPipe (on page 2289)

V8i (release 20.07.05)

The latest What's New document for STAAD.Pro V8i contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (2007 build 04). This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-05.1 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2007-05.1.1 Geometric Nonlinear Analysis

The range of analysis options has been supplemented with a new solution to account for nonlinear effects of moderate displacement and small strain. This solution holds for where the element distortion is small and small rotations are assumed.

Note: The nonlinear analysis command is available with the Advanced Analysis license.

Related Links
- TR.37.8 Geometric Nonlinear Analysis (on page 2830)
- A. To specify a nonlinear analysis (on page 930)

AD.2007-05.1.2 IS 800:2007 Steel Design

The Indian Bureau of Standards has released a new version of the design code for the design of steel structures, known as IS 800:2007. This replaces the previous version of the code (also supported in STAAD.Pro), IS 800:1984, which was versified again in 1998. This is a new approach to steel design and is based on the limit state design rather than a stress based design of the old code.
The Steel Design section has been enhanced to include design per IS 800:2007. Both section checking and selection routines are supported.

The design will follow the same process as used by all other steel design codes currently available in STAAD.Pro. The design of each member is controlled through a set of parameters that have been added to the GUI Steel Design Dialog under the title **IS 800:2007**.

The design engine will allow all standard section database sections and User Table sections to be designed.

The design process followed:

1. Check slenderness  
2. Check classification  
3. Check tension forces  
4. Check compression forces  
5. Check bending forces  
6. Check interaction

The results will be:

1. output to the ANL file  
2. available in the member query  
3. available in the post processing mode in the Design Results table

**Note:** *Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used).*

**Related Links**

- **D8.E.4 Design Parameters** (on page 1951)

**AD.2007-05.1.3 Eurocode 3 Includes National Annex**

A number of countries that have signed up to the replace their current steel design standards with the Eurocode, EN 1993-1-1:2005, known commonly as Eurocode 3, have published their National Annex documents. These documents make small changes to the base document and STAAD.Pro has been updated to incorporate some of these National Annex documents. Currently, the Dutch and Norwegian National Annexes have been added to the STAAD.Pro engine.

A new parameter, NA, that sets the default material gamma factors and any additional changes outlined in the country specific National Annex such as specific equations or methods.

The output file printout has been updated to indicate which National Annex (if any) has been used in a code check / select process. (For all TRACK settings)

**To specify checks per a National Annex**

In order to include additional check specified by a National Annex, do the following:

1. From the Modeling mode Design | Steel page, select **EN 1993-1-1:2005** from the current code list.
2. Click Define Parameters to launch the Design Parameters dialog.
3. Select the parameter NA from the list.
4. Select the radio button for the National Annex you wish to use; or leave as Basic in order to use EC3 without additional checks.

5. Click **Add** to add the NA parameter to the code check.

6. Click **Close** to dismiss the dialog once parameter definitions are complete.

A design performed to the new Eurocode 3 National Annex is displayed in the output file (*.ANL) with the following header, in addition to the base EC3 output:
### What's New?

**STAAD.Pro V8i**

---

**ALL UNITS ARE - KN METRE (UNLESS OTHERWISE NOTED)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>EC-6.2.5</td>
<td>0.723</td>
<td>1</td>
<td>0.0</td>
<td>50.0</td>
<td>0.0</td>
</tr>
<tr>
<td>EC-6.2.6-(Y)</td>
<td>0.284</td>
<td>1</td>
<td>0.0</td>
<td>50.0</td>
<td>0.0</td>
</tr>
<tr>
<td>EC-6.3.2 ITB</td>
<td>0.757</td>
<td>1</td>
<td>0.0</td>
<td>50.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**ADDITIONAL CHECKS AS PER NATIONAL ANNEX [NEN-EN 1993-1-1/NB] (units- kN,m):**

<table>
<thead>
<tr>
<th>EC CLAUSE</th>
<th>NA-CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>EC-6.2.8-(Y)</td>
<td>NEN-6770-Eq.11.3.1</td>
<td>0.689</td>
<td>1</td>
<td>0.0</td>
<td>50.0</td>
<td>0.0</td>
</tr>
<tr>
<td>EC-6.2.10(Y)</td>
<td>NEN-6770-Eq.11.3.1</td>
<td>0.689</td>
<td>1</td>
<td>0.0</td>
<td>50.0</td>
<td>0.0</td>
</tr>
<tr>
<td>EC-6.2.10(C)</td>
<td>NEN-6770-Eq.11.3-31</td>
<td>0.550</td>
<td>1</td>
<td>0.0</td>
<td>50.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.

---

**Note:** The previous, development edition of Eurocode 3 is included as the Code EN3 DD.

---

**AD.2007-05.1.4 Eurocode 8**

Eurocode 8: Part 1 [EN 1998-1-1:2004] contains specific requirements and recommendations for building structures that are to be constructed in seismic regions. Essentially, these fundamental requirements have been provided to ensure that the structures can sustain the seismic loads without collapse and also – where required – avoid suffering unacceptable damage and can continue to function after an exposure to a seismic event.

As with all Eurocodes, a National Annex Document should accompany the use of Eurocode 8 in each of the European nations.

**Related Links**

- [D. To perform seismic design and detailing per ECB](on page 1018)

---

**AD.2007-05.1.5 AIJ Concrete Design Update**

An old version of Japanese concrete code based on the AIJ standard for structural calculation of Reinforced Concrete Structures (1985 edition) was previously implemented in STAAD.Pro batch mode design. Recently, critical errors in the column design and beam design were discovered and corrected. In addition, the AIJ concrete design has been updated to incorporate the latest AIJ standard for structural calculation of Reinforced Concrete Structures (1991 edition).
Generally, this implementation will be invisible to you. The 1985 edition of the AIJ has been completely replaced by the 1991 edition for the design of concrete members per the AIJ. The existing commands as described in D9.A. Japanese Codes - Concrete Design per 1991 AIJ (on page 1969) are applicable to the 1991 edition.

**AD.2007-05.2 Features Affecting the Concrete Design Mode**

An existing feature has been modified in the RC Designer section of the program, also known as the Concrete Design mode. This is explained in the following pages.

**AD.2007-05.2.1 RC Designer Member and Envelope Import**

The ability to create physical members in STAAD.Pro has been introduced since the original implementation of the RC Designer. Previously these members did not get carried through into the RC Designer, thus requiring the extra step of re-creating these members, and possible confusion if the same member numbers were not used.

Similarly, load case/combination envelopes were not carried through into the RC Designer, requiring extra care in identifying which load cases were being considered when examining the output.

The physical member definitions are now passed through to the RC Designer along with the other structural data from STAAD.Pro. Envelope definitions are also carried through and examined to see if they contain any dynamic loading for which it may be inappropriate to use the analysis results directly.

Generally, this implementation is invisible to the user. Physical member assignments made in the STAAD.Pro Modeling mode will be carried over to the RC Designer (Concrete Design mode). If existing designs are re-examined, the user may be warned if member numbers or constitution (in terms of beam elements included) differ between the STAAD.Pro Modeling mode and the RC Designer definitions, and the option is given to delete the RC Designer members with their results. For envelopes, the user is prompted if the envelope numbers or constitution differ between RC Designer and STAAD.Pro Modeling mode.

Additionally, a warning may be presented to the user if any envelope incorporates a dynamic load case. This is because the dynamic load cases in STAAD.Pro only produce positive values for moments and forces, even though it is understood that the loading actually reverses. Concrete design based on an envelope of positive forces only would be seriously under-reinforced.

**V8i (release 20.07.04)**

The Software Release Report for STAAD.Pro V8i contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 03. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

**AD.2007-04.0 New Features Affecting the General Program**

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.
AD.2007-04.0.1 ProjectWise Integration

ProjectWise is an engineering project team collaboration system which is used to help teams improve quality, reduce rework, and meet project deadlines. One of the major pieces of functionality provided by ProjectWise is an Integration Server which allows data to be managed and shared across a distributed enterprise.

STAAD.Pro has been enhanced so that the model STD data file can be managed on a ProjectWise server. Installation and management of a ProjectWise server is beyond the scope of this document and should be obtained from the ProjectWise installation.

A local ProjectWise client should be installed which allows access to ProjectWise repositories.

When STAAD.Pro is launched, the option to open and check out a STAAD.Pro STD file from a ProjectWise repository is made available from the Project Tasks on the Start Page thus:

This is also available from the File menu while still on the Start Page prior to opening a model:
If a suitable ProjectWise client is not installed, then the link on the Start Page is shown as unavailable with a red line through the icon thus:

![Project Tasks]

As authentication is required to access files stored on a ProjectWise repository, a login dialog allows the required details to be entered either with specific user credentials or by using the current windows login credentials thus:

![ProjectWise Log in]

Files that are accessed from a ProjectWise server are “Checked Out” and stored locally during the STAAD.Pro session until the file is closed and then it is returned to the server.

The first time that a successful link to a ProjectWise server is established, a location in which check out files are to be stored locally and additionally, where all the auxiliary data files are stored whilst STAAD is running is required. Afterwards and on all future occasions, the ProjectWise open dialog presented is then presented where the repository can be navigated and filtered as defined in the ProjectWise documentation.
Note the significance of the icons next to the STAAD filenames. These indicate the status of the file such as the current document, checked out to you, or locked as checked out to some other user. Refer to the ProjectWise documentation for a full description of each icon.

With a file checked out and loaded in STAAD.Pro, it is possible to see the ProjectWise Properties, by selecting the option from the ProjectWise toolbar or File menu:

If working on a STAAD model that has not originated from a ProjectWise server (e.g., starting a new file) and it has been decided that it needs to be added to a repository, then at any time whilst working in the STAAD.Pro environment, clicking on the toolbar option Add to ProjectWise server, or equivalent File > ProjectWise menu option will launch the following dialog:
Selecting the option **No Wizard** offers the following dialog into which the file details can be entered. 

Clicking on Save, adds this model to the repository, but indicates its status as checked out until the file is closed in STAAD.Pro and the model checked back in.

Alternatively, by selecting the **Advanced Wizard** option the data needed to define the ProjectWise data file is presented in the following four steps:

1. Advanced Wizard
2. Select target Folder

3. Document Properties
4. Create the document

Creates the document and marks it as checked out until the file is closed and checked in to the server.

Four integration functionalities have been added. These are:

- Open a STAAD model from a ProjectWise repository.
- Save a local STAAD model into a ProjectWise repository.
- Update an existing model from ProjectWise.
• Review model properties (meta-data) which has been opened from a ProjectWise repository.

Note that access to all of these functionalities is available from ProjectWise sub-menu under the general File menu described below.

**Saving changes of a Checked Out model Back on the Server**

When a model is checked out from a ProjectWise server, selecting Save or Save As, only maintains a local copy of the model. There are two methods available to update a checked out model. Firstly, during a STAAD.Pro session, it is possible at any time to save any changes back on the server by selecting the Update Server Copy icon from the ProjectWise toolbar or from the File > ProjectWise menu.

First save updates to the file locally. If not, then this will be prompted. Then the following dialog is displayed which allows a comment to be added to this model, thus:

![Update Server Copy dialog](image)

The file remains checked out and can be continued to be worked on.

The second method is automatically generated when a checked out model is closed. This launches the Check In dialog that displays the file(s) that are to be checked in and provides four actions:
Check In
This copies the local version of the STAAD model back to the server and releases it so that it is available for others to modify. If the option Create New Version is selected, then a new copy of the file is created on the server which becomes the current version of the model. The status of the checked out model is changed to read only on the server and can be used as a reference to a stage in the development of the model.

Update Server Copy
This updates the model on the server with the current local model, but does not change its status which remains as checked out.

Free
This changes the status of the model as checked in which allows other users to take control and check out the model, but does not update the server model. Thus local changes will be discarded.

Cancel
This cancels any change to the status of the model or the model itself on the server.

Menu
The commands that drive the ProjectWise integration are defined in the main File menu thus:

File > ProjectWise
The Add... command is available when using a locally opened (not checked out from ProjectWise), which allows the current model to be saved into a ProjectWise repository.
The Open... command is available when a suitable ProjectWise client has been installed to allow access to a repository from which to check out a STD file.
The Update Server Copy command is available when working on a checked out file and the changes made on the model can be
Toolbar

A new toolbar named ProjectWise has been added that duplicates the commands from the File > ProjectWise menu, thus:

Notes

1. For more details on ProjectWise refer to the ProjectWise client installation documentation.
2. This functionality requires access to a version V8i or greater of ProjectWise.

AD.2007-04.0.2 CIS/2 Update

The STAAD.Pro tool to import and export models with the CIS/2 translator has been enhanced to work with international models and transferring models into 3D modeling such as SmartPlant 3D.

The CIS/2 (CimSteel Integration Standard, Version 2) allows for the transfer of steel models using a prescribed data standard in the STEP (Part 21) format. These files can contain different models including analysis models. The STAAD.Pro CIS/2 translator only operates on the analysis model within the file. While STAAD.Pro has supported a wide range of the steel sections that this standard can support, this enhancement allows a far greater range of sections to be imported/exported with this tool.

Import

The CIS/2 import can be initiated after starting a new model and before creating any model data, selecting the menu option, File > Import...

Selecting the CIS/2 option and clicking on the “Import” button allows selection of a suitable STEP file. This file is a text file and will be similar to the following extract:

```
ISO-10303-21;
HEADER;
/* Generated by software containing ST-Developer */
/* from STEP Tools, Inc. (www.steptools.com) */
FILE_DESCRIPTION(
    /* description */ ('CIS2 Export File'),
   */
```
Once a file has been selected, the data is ready to be imported with the CIS/2 import tool thus:

Clicking the Import button then runs the model import where the CIS/2 file is processed and the analysis model data is extracted to form the STAAD.Pro model.
Note that as the CIS/2 file is being processed, a log file which identifies the data that has been utilized in the STAAD model is produced and displayed which can be saved as a text file for future reference.

**Export**

The CIS/2 Export is available for any model that has been created as an option in the menu item File>Export...

Selecting the CIS/2 option and clicking on the “Export” button allows the definition of a suitable STEP filename and folder to locate it. Once again the CIS/2 tool is presented, this time with an Export Model button at the bottom which when clicked creates the STP file.
Once again, when the export has completed, the user can save the log file which is produced as the model is converted into the STEP format.

Enhancements

The STAAD.Pro CIS/2 import now recognizes sections defined from standard databases such as those defined in Japanese, British, Indian, Australian and European tables.

The import/export has been enhanced to support the ability of STAAD.Pro to create double sections, such as back to back angles and channels or double wide flange sections. Additionally, it now supports the creation of T sections which are defined in STAAD.Pro as a wide flange section that is split at mid height.
Although CIS/2 has been developed for the processing of steel models, the STAAD.Pro translator will now support the transfer of prismatic properties normally associated with concrete sections. This means that sections defined as PRISMATIC in a STAAD.Pro model will be included in exported STP file and can be imported if they exist.

Related Links
- [I. To import a CIS/2 file](on page 2287)
- [I. To export to a CIS/2 file](on page 2289)

AD.2007-04.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.

AD.2007-04.1.1 ASME NF Steel Design Codes

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented and the steel design page has been updated to allow the design parameters to be defined and assigned.

The design requirements for the following years have been added:
- 1974
- 1977
- 1989
- 1998

Post analysis steel design code checking requirements for the required ASME NF code can be selected by entering the Design | Steel page and setting the ASME code / year in the Current Code option in the Steel Design - Whole Structure dialog:

The method for selecting design parameters and assigning them to the members to change them from the default values is exactly the same as for all other steel design codes. Additionally, choosing members that are to be checked or selected for maximum utilization follows exactly the same method as for all other steel design codes.

For more information on the technical requirements of this design code, including the full set of parameters and default values, see the new section in the D1.L. American Codes - Steel Design per ASME NF Codes (on page 1588).
Note: In order to run a design check to any of the ASME NF design codes, then access to a STAAD Nuclear Code pack will be required.

AD.2007-04.1.2 Floor Response Spectrum

A new dynamic feature has been added that allows the extraction of a response spectrum from a collection of nodes that constitute a floor when subjected to a time history loading. This information can then be used in conjunction with equipment that will be supported by these floors and is often required by the equipment manufacturers.

Users will require a license for the advanced analysis module to access this feature.

The required commands (see AD.2007-04.2.3 Floor Response Spectrum (on page 206) for more information) can be entered graphically after adding the analysis command by selecting the new analysis sheet Generate Floor Spectrum thus:

![Generate Floor Spectrum sheet](image.png)

This command should follow immediately the definition of the analysis and will require defined groups of nodes which need to be defined first.

The following displays the layout of this sheet:
Note: Each line of settings constitutes one floor which can have one or more floor groups assigned. The resulting response spectra will be based on the collective responses of all the nodes in the selected groups.

Once the required options have been set, click on the Add button to add the command set to the model which should appear in the Analysis Window thus:
Once the command has been added and the file saved, the analysis can be run which will generate a new sub-page in the Post-Processing Mode in the Dynamics Page called Floor Spectrum:
Entering this page, STAAD.Pro will display the floor spectrum thus:

To change graphs to that of another selection of node groups or damping ratio, then select the required set from the drop list in the Response Spectrum Table.

The graph is initially set to display the results on a log/log graph. This can be changed to a linear graph by right clicking on the graph and selecting the "Linear Graph" option. Additionally, the calculated points on the graph can be added by again right clicking on the graph and selecting the option ‘Show Points’.
The data that has defined the graph can also be exported to a text file and used in a third party application by right clicking on the graph and selecting the option “Save Data in Text File…”

Related Links
- TR.37.10 Floor Spectrum Command (on page 2833)
- A. To generate a floor spectrum (on page 935)
- P. To display floor spectrum results (on page 2218)

AD.2007-04.1.3 Russian Wind Loading

The wind loading as defined in the design code SNiP 2.01.07–85 “Loads and Actions” added in this version of STAAD.Pro can be added graphically to the model by modifications in the Loads Page.

There are two areas where the graphical user interface has been updated:

**Wind Load Definition dialog**

The dialog has been updated to allow the entry of the required parameters for a Russian Wind Load definition thus:-

![Wind Load Definition dialog](image)

This will create a definition which can be added to a wind load case.
Wind Load

Note: The first load case which has a Russian Wind Load command added to it will consider all other loads defined in it as the masses to be considered for calculating the dynamic effect which is required by this command.

For technical details of this wind loading see section AD.2007-04.2.2 Russian Wind Loading (on page 205).

AD.2007-04.1.4 Additional Standard Profile Databases

An additional Australian cold formed database has been provided to complement the cold formed sections databases currently provided.

The following database and tables have been added from OneSteel. Duragal®, Galtube® and Tubeline®

Australian Cold Formed Steel Hollow Sections

Circular Hollow Sections:

- Galtube Plus®, 26.9mm to 76.1 mm diameter
- Tubeline, grades C250L0 (AS1163) and 350L0 (AS1163), 26.9mm to 457.0mm
Rectangular Hollow Sections

- Duragal®, grade C450L0 (AS1163), 50x20mm to 150x50mm
- Galtube Plus®, grade C350L0, 50x20mm to 75x25mm
- Tubeline®, grade C350L0 (AS1163), 50x20mm to 250x150mm

Square Hollow Sections

- Duragal®, grade C450L0 (AS1163), 20x20mm to 100x100mm
- Galtube Plus®, grade C350L0, 20x20mm to 65x65mm
- Tubeline®, grade C350L0 (AS1163), 13x13mm to 250x250mm

AD.2007-04.2 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2007-04.2.1 ASME NF

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented and the steel design page has been updated to allow the design parameters to be defined and assigned.

The design requirements for the following years have been added:

- 1974
- 1977
- 1989
- 1998

For the list of parameters and commands including the default values, please refer to the D1.L. American Codes - Steel Design per ASME NF Codes (on page 1588).

For each steel member that is checked, the following checks are performed according to the clauses for that year:

1. Slenderness check
2. Tension
3. Compression
4. Bending (a) About major axis, (b) About major axis
5. Shear
6. Combined Stresses

AD.2007-04.2.2 Russian Wind Loading

The wind loading commands have been enhanced to allow the creation of wind loading as defined in the in Russia by the design code SNIP 2.01.07–85 Loads and Actions.
The basic quantity in the wind loading is the characteristic (normative in Russian terminology) wind pressure. The reference wind velocity pressure corresponds to a 10-minute time-averaged velocity pressure at 10-metres height in a flat terrain, based on a 5-year return period. This wind pressure is the static component of the wind load. The total wind pressure consists of static and fluctuating components. If the structure is sufficiently flexible, according to the code provisions, the dynamic structural response to the fluctuating wind component must be taken into account.

The updated wind loading commands automatically perform both the aerodynamic and structural load analysis of vibration-susceptible buildings and structures.

The wind loading commands have been updated to support the wind loading as defined in the Russian design code. This requires the creation of the following commands.

There are three parts to creating a Russian wind load on a STAAD Model.
1. Definition of the wind load requirements
2. Application of the wind load definition within a load case
3. Cut-off frequency or mode shape

Related Links
- TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2539)
- TR.31.3 Definition of Wind Load (on page 2623)
- TR.32.12.3 Generation of Wind Loads (on page 2779)
- M. To add a SNiP wind load definition (on page 837)

AD.2007-04.2.3 Floor Response Spectrum

The following commands have been added in order to allow the response spectrum of floors to be extracted from a time history analysis.

This command is used to specify the calculation of floor and/or joint spectra from time history results. The Floor Response Spectrum command must immediately follow an analysis command. That analysis can only contain a single time history load case.

Related Links
- TR.37.10 Floor Spectrum Command (on page 2833)
- A. To generate a floor spectrum (on page 935)

AD.2007-04.3 Features Affecting the Post-Processing (Results Mode)

Several new features have been added and existing features have been modified in the post-processing section of the program, also known as the Results Mode. These are explained in the following pages.

AD.2007-04.3.1 Time History Animation

In order to visualize the displacement that occurs on a model during the application of a time history load, a new toolbar icon has been added which allows the displacement at a specific time instance to be displayed.
The results toolbar has been updated to include a new option which is activated when the displacement diagram icon has been clicked and the current load case contains time history loading:

For a time history load case the displacement that is displayed will be defined by the time instance entered in the above edit box. Alternatively the time can be set from a sliding scale by clicking on the button with the down arrow icon which displays the following option setting:

The slider scale is based on the overall time for which the time history analysis has been performed. The displacement is produced at the time at which the slider arrow is dragged to and the mouse button released.

If the **Apply Immediately** option is selected, then the application will attempt to render the displacement diagram dynamically as the slider is dragged up and down the scale of time. For large models this may prove to be too demanding on the graphics system and left un-checked which means that the displacement diagram will be produced at the time step at which the slider arrow is released.

Note that the displacement diagram will be set to the scale as defined in the Scales sheet in the **Structure Diagrams** dialog box.

Additionally note that it is possible to view the time v displacement of individual nodes by clicking on the Dynamics>Time-Disp in the post processing mode which is only available for models that include time history.
**AD.2007-04.3.2 Enhanced Plate Stress Results**

In order to provide additional understanding of stress distribution in finite element models, STAAD.Pro has expanded the sets of results that can be reported for each element both in the Plate Centre Stress Table and graphically using the Plate Stress Contour.

To view the results data of plate elements, enter the Post Processing (Results) Mode, and click on the Plate Contour Page on the left menu.

**Plate Centre Stress Table**

The new Combined Stresses sheet in the Plate Centre Stress table provides resolved stresses for the top (positive local Z) and bottom (negative local Z axis) surface for each plate element, referred to as the Top Combined Stresses and Bottom Combined Stresses respectively.

The combined stresses are calculated thus:

**Top:**

\[
\begin{align*}
S_{X_{\text{top}}} & = SX + \frac{MX}{S} \\
S_{Y_{\text{top}}} & = SY + \frac{MY}{S} \\
S_{XY_{\text{top}}} & = SXY + \frac{MXY}{S}
\end{align*}
\]

**Bottom:**

\[
\begin{align*}
S_{X_{\text{bottom}}} & = SX + \frac{MX}{S} \\
S_{Y_{\text{bottom}}} & = SY + \frac{MY}{S} \\
S_{XY_{\text{bottom}}} & = SXY - \frac{MXY}{S}
\end{align*}
\]

where

\[
\begin{align*}
S & = \frac{t^2}{6} \ t \\
tS & = \text{average plate thickness}
\end{align*}
\]

**Plate Stress Contour**

The Plate Stress Contour sheet of the Diagrams dialog has been enhanced to allow visualization of these stresses. The six new stress results are available in the Stress Type pull down menu thus:
What's New?

STAAD.Pro V8i User Manual
STAAD.Pro 2007 Release Reports

This section of the Software Release Report contains detailed information on additions and changes that have been implemented in builds of STAAD.Pro 2007 since the release of STAAD.Pro 2006 build 1005. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

STAAD.Pro 2007 Build 03 Release Report

The Software Release Report for STAAD.Pro 2007 Build 03 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 Build 05. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

STAAD.Pro 2007 Build 02 was a maintenance update and only contained a number of minor updates which are listed in the Revision History text file.

AD.2007-03.0 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

AD.2007-03.0.1 RSS Feed added to the Start Page

The STAAD.Pro Start Page has been updated to include a new section which has been configured to display the most current information about STAAD and Bentley that will be of use to you, such as program updates, seminars, and training courses. This has been done by adding an RSS (Really Simple Syndication) reader on the application front screen.
Each news item is identified with a title which is a link to a website which can be clicked on and will launch your web browser and load that website and a brief summary of the item.

The item is categorized with one of the following five categories:

- **Important**
- **Bentley General**
- **Release**
- **Educational**
- **News**
Additionally, each item is marked for specific regions thus:

To configure which messages are displayed and for which region, click on the **Configuration...** link on the Start Page. Select the RSS Feed and click in the categories and regions to remove those that are not required.

**Note:** Items that have been classed as Important and for All Regions cannot be disabled, but other issues can be by un-checking the boxes next to them.
AD.2007-03.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.

---

AD.2007-03.1.1 New Meshing Options

The Parametric Models section STAAD.Pro has been enhanced with additional meshing options to provide three methods for creating a finite element mesh. These methods are classified as **Basic**, **Standard**, and **Advanced**.

The original content of this topic has been superceded by content in the main help. Please refer to the Mesh Parameters dialog (on page 2912) in the User Interface help for current details.

---

AD.2007-03.1.2 Enhanced Automatic Load Combination Generation

The Automatic load combination generator has been enhanced with two options that define whether notional loads should be included in the combinations and how combinations are created to allow a far greater range of combination conditions to be considered.

The two new options are displayed on the top rows of the dialog box that is opened from the Pre-Processor menu item: **Commands > Loading > Edit Auto Rules**

---

**Related Links**

- **Edit Load Rules for Auto Load Combination Generator dialog** (on page 3035)
- **M. To define automatic load combination rules** (on page 867)
AD.2007-03.1.3 Generation of Primary Load Cases Using Repeat Load Commands

With the large number of combinations that are required by design codes, STAAD.Pro introduced a tool to allow the combinations to be created automatically based on a rule set. This functionality has been enhanced to allow the automatic creation of equivalent combinations, but using primary load cases and the REPEAT command.

The use of the COMBINATION command to obtain results for a collection of load cases that occur together is entirely valid when using first order analysis and models that are not dependant upon the forces in them (such as tension only members or one-way supports). In these cases, the law of superposition allows the results of each load case to be multiplied by a scaling factor and added together. However, for models where this does not apply, such as using a P-DELTA analysis, TENSION ONLY members or ONE-WAY supports, then the law of superposition does not necessarily apply. For this reason, STAAD.Pro has the ability to create primary load cases which replicate the effect of a combination, but in a primary load case. This means that the analysis will solve the load case by using the applied loading rather than simply adding the results. This is achieved in a primary load case by using the REPEAT command.

For full details of the REPEAT command see TR.32.11 Repeat Load Specification (on page 2770).

There are three parts to automatically creating primary load cases equivalent to combinations.

1. Define the rules that determine how load case types are to combined
2. Define the individual primary load cases classified with their type
3. Create the combinations.

By selecting the option Create Repeat Load Cases all the load case combinations in the Selected Load Combinations list will be created into primary load cases:

Note the difference in the Load & Definition dialog when comparing a generation of Combination load cases rather than Primary load cases with REPEAT commands:
What's New?

STAAD.Pro 2007 Release Reports

**Related Links**

- *Edit Load Rules for Auto Load Combination Generator dialog* (on page 3035)
- *M. To define automatic load combination rules* (on page 867)

**AD.2007-03.1.4 Design Code List**

The list of supported design codes in the design dialog boxes has been updated to represent the actual document rather than the country where this is possible.

The list of codes displayed in the Steel, Concrete, Timber, and Shearwall Design dialog boxes has been updated such as here for the Steel **Design** dialog box:
Note that there is no change to the input or output generated from the design unless noted in the section, AD. 2007-03.2 Features Affecting the Analysis and Design Engine (on page 223).

AD.2007-03.1.5 Additional Standard Profile Databases

An additional Japanese cold formed database has been provided to complement the cold formed sections databases currently provided.

The following database and tables are available.

*Japanese Cold-Formed Steel*

- Box Column Press
- Box Column Press (Tough)
- Box Column Roll
AD.2007-03.1.6 IBC 2006

The static equivalent method for performing dynamic analysis as per the IBC 2006 code has been implemented. This follows the principals and methods as in the IBC 2000 and IBC 2003 codes previously implemented.

When the **General | Load** Page is selected, the right hand side of the screen will display the following if no load cases exist in the model.

![Image](image1.png)

**Definitions** contains the options through which one creates the **Define** block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases.

When the tree view is expanded, it will display thus:

![Image](image2.png)

To define the definition of the IBC 2006 select the Seismic Definitions and click on Add. As with other seismic definitions, there are three parts to the exercise,

1. Defining the code and parameters
2. Defining the weights to be considered
3. Include the definition in a primary load case
In the dialog box that is displayed, select **IBC 2006** from the drop-down list.

In this dialog box, the required data is entered into the parameters as described below.

There is an option to **Include Accidental Load**, if this is selected the analysis will include an additional accidental torsion component as described in section 12.8.4.2 of ASCE 7-05.

Refer to the Add New Seismic Definitions dialog (on page 3044) and TR.31.2.12 IBC 2006/2009 Seismic Load Definition (on page 2592) for details.

After specifying the values for the parameters, click on the **Add** button.

Note that the **Load** dialog box has been updated with the new command.
As described earlier, the second part of the process is to identify the structural weights that should be considered. These can be added once the defined parameters have been created by clicking on the Add button. The dialog box is updated to display the various weights that can be added thus:

**Selfweight**

This is the selfweight of the structure.

**Joint Weights**

These are the concentrated weights acting at one or more joints.

**Member Weights**

Distributed and concentrated weights acting on member spans are specified through this option. After clicking the Member weight button, the Member weight dialog box appears, as shown below.
Select the Concentrated or Uniform load type from the Loading Type drop-down list. Enter the intensity of the distributed weight or magnitude of the concentrated weight as the case may be, along with the location of the load.

**Element Weights**

If the structural model consists of plate elements representing entities like floor slabs, the pressure loads on those slabs can be considered for weights calculation for lateral load generation per UBC/IBC/other codes. This is done with the help of the Element Weights option. Its parameters include the magnitude of the uniform pressure, and the elements they are applied on. Since it is a weight, it is a quantity without a sign.

**Reference Load**

Reference Load cases which are described in AD.2007-1001.1.12 of the STAAD.Pro 2007 Software Release Report can be referred to using this option. Loads which are specified under Reference Loads can be used as weights for IBC.

**Floor Weights**

In many situations, a user may decide not to include the structural slabs in his/her analytical model. Hence, the model may be solely the skeleton framing system consisting of the beams, columns and bracing members.

Under these circumstances, the loads which act on the slab can no longer be applied on the structure using the ELEMENT PRESSURE options. This is because there are no elements to represent the slab. So, an alternative is to apply the load using the FLOOR LOAD option. It is described in detail in TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664).

Within a UBC/IBC/other codes definition, the FLOOR WEIGHT is the counterpart for the FLOOR LOAD just as MEMBER WEIGHT is the counterpart for a MEMBER LOAD, and an ELEMENT WEIGHT is the counterpart for an ELEMENT LOAD.

**Note:** Its parameters are hence very similar to what are found in a normal FLOOR LOAD definition. XRANGE, YRANGE and ZRANGE options allow the user to narrow in on panels at specific regions of the building. The pressure value is provided as a quantity without sign because it is contributing to the overall weight - a numerically positive term.
Once the seismic definition, including weights, has been specified, it should be included in a one or more primary load cases.

Select **Load Cases Details** and click on **Add**.

In the dialog box that is displayed, provide a **Title**, select the loading type as 'seismic' (although this is not strictly required, just useful for future reference) and click on **Add**.

More load cases can be added in this manner.

To add load items to our first load case, keep the expression **1: SEISMIC IN THE X DIRECTION** highlighted and click on the **Add** button.
Enter the **Factor**, **Direction**, etc. and click on **Add**.

The **Load** dialog box will display the new load item.
We can continue adding other load items to this load case in a similar fashion.

For more information on other UBC and IBC load definitions, see TR.31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2544). To find more information about including the seismic loads in a primary load case, see TR.32.12 Generation of Loads (on page 2771).

**AD.2007-03.2 Features Affecting the Analysis and Design Engine**

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

**AD.2007-03.2.1 Selfweight Command with a Member List**

When considering stage construction, it is occasionally necessary to create load cases which include the effects only of the additional parts of structure. In order to do this the SELFWEIGHT command has been enhanced to allow a member list to be added which will be processed and only selfweight on that list will be considered.

This command may be used to calculate and apply the SELFWEIGHT of the structure for analysis. This allows you to enter body weight of the structural components. In other words, it allows different parts of the structure to be excited with different accelerations.

Refer to TR.32.9 Selfweight (on page 2685) for details on using member lists with the Selfweight command.
**Example**

```plaintext
LOAD 1 DEAD AND LIVE LOAD
SELF WEIGHT X 1.4 LIST 4 TO 10
SELF X 1.0 _PLATEGRP1
SELF
SELF Y -1.4
SELF X 1.4 ALL
SELF WEIGHT Z 1.4 _SOLIDGRP1
SELF Y -1.4 YR 10.5 10.51
SELF Y -1.4 X
```

In above example, the first specification includes factored weight of member (/plate/solid) 4 through 10 along global X direction and the second includes weight of all members (/plates/solids) associated with group _PLATEGRP1. The third command includes weight of all structural components towards gravity direction (SELF WEIGHT Y -1.0).

SELFWEIGHT specification for definition of static method of seismic load generator also is capable of accepting list. Only there is no direction specification for this command.

Similarly, the SELFWEIGHT specification when used in a Reference Load is also capable of handling a list.

---

**AD.2007-03.2.2 Direct Analysis**

The AISC 360-05 Appendix 7 describes a method of analysis, called Direct Analysis, which accounts for the second order effects resulting from deformation in the structure due to applied loading, imperfections and reduced bending stiffness of members due to the presence of axial load.

This is a non-linear iterative analysis as the stiffness of the members is dependent upon the forces generated by the load. The analysis will iterate, in each step changing the member characteristics until the maximum change in any Tau-b is less than the tau_tolerance, If the maximum change in any Tau-b is less than 100*tau_tolerance and the maximum change in any displacement degree of freedom is less than the disp_tolerance; then the solution has converged for this case.

There are two steps involved in setting up a Direct Analysis.

1. Specify the definition with a DEFINE DIRECT command.
2. Specify a direct analysis method with the command PERFORM DIRECT ANALYSIS

**Note:** Like all other analysis methods, by specifying the direct analysis parameters and only including a PERFORM ANALYSIS command, will result in only a first order elastic analysis, not a direct analysis to be performed.

**Related Links**

- [M. To define direct analysis parameters](on page 852)
- [A. To specify a direct analysis](on page 929)

---

**AD.2007-03.2.3 Notional Loads**

A number of design codes require that a notional load be considered. Typically this is defined as a percentage of the gravity loads. What has been added to STAAD.Pro is an enhancement on the REPEAT LOADS command so
that Primary and /or Reference load cases can be selected and a percentage of them can be applied in the appropriate global direction.

The addition of the Notional Loads command in the GUI is as an additional option on the Repeat Load option of the New Load Items, thus:

Both Principal and Reference load cases can be selected and moved into the Notional Load Definition where the required factor and direction can be specified.

The notional loads are calculated and applied as joint loads. Note that the actual values of the applied loading are not displayed in the GUI until after the analysis has been performed.
AD.2007-03.2.4 STP Parameter Added to AISC 360-05 Design

The AISC 360-05 design of wide flange depends on whether the section is formed from a rolled process or from plates welded together.

The parameter STP has been added to the list of parameters to indicate whether the section is built up from welded plates or from a rolling process. This is then used to determine the appropriate slenderness ratio as defined in table B44.1 from AISC 360-05. This parameter takes the following values:

1. Rolled Section
2. Built-up section

The default value is 1.0

Related Links
- D1.A.6 Design Parameters (on page 1378)

AD.2007-03.2.5 Updated and Additional Standard Steel Grades in Eurocode 3

The standard steel grades are referenced in the Eurocode 3 have been provided as standard steel grades in order that the appropriate buckling curves can be selected.

The steel grade can be specified using the parameter SGR which accept values of 0 to 5. This translates to the following:

<table>
<thead>
<tr>
<th>Grade</th>
<th>SGR</th>
</tr>
</thead>
<tbody>
<tr>
<td>S 235</td>
<td>0 (default)</td>
</tr>
<tr>
<td>S 275</td>
<td>1</td>
</tr>
<tr>
<td>S 355</td>
<td>2</td>
</tr>
<tr>
<td>S 450</td>
<td>3</td>
</tr>
<tr>
<td>S 460</td>
<td>4</td>
</tr>
</tbody>
</table>

Note: If a user-defined yield stress PY is specified, the buckling curve will still be determined from the SGR setting (i.e., if not specified, it will be based on the default SGR parameter). Therefore, if setting a user PY, this should be preceded in the file by the appropriate steel grade command.

Related Links
- D5.C.6 Design Parameters (on page 1776)

AD.2007-03.2.6 Russian Response Spectrum

The Response Spectrum functionality has been enhanced with the ability to create response spectra as defined in the SNiP II-7-81.

To create a response spectrum as defined in the SNiP code, enter the General | Load & Definition Page and create or select the load case which is to be used. Click on the Add... button and select the Response Spectra option from the left panel.
Enter the required data in the provided fields and click on the Add button to include the command in the selected load case of the data file.

**Related Links**
- **TR.32.10.1.11 Response Spectrum Specification per SNiP II-7-81** (on page 2759)

**AD.2007-03.2.7 Eurocode 3 Updated to Support Design of Slender Box Sections**

Both the ENV and DD versions of Eurocode 3 steel design need to design box sections that have slender elements.
Members that are specified with steel hollow box section properties that contain slender elements can now be designed to Eurocode 3. Previously, these members would not be designed and result in a warning in the output stipulating that slender members are limited to I section properties only.

**AD.2007-03.2.8 AIJ 2005 Steel Design**

The AIJ has updated the Japanese steel design code, AIJ 2005. This design code has been implemented and complements the existing Japanese steel design code AIJ 2002. The changes are minor and the code checking process remains the same as the previous AIJ design.

The design parameters are the same and apply as defined for the current AIJ, the only exception, however, is the **Current Code** parameter which must be set to **AIJ 2005**. This setting is available from the GUI list of codes.

![Steel Design - Whole Structure](Image)

Note that a design performed to the new AIJ 2005 standard is displayed in the output file (*.ANL) with the following header:

```
STAAD.PRO CODE CHECKING - (   AIJ 2005     )
********************************************************************
```

The equivalent header for a code check (or member selection) to the older standard is displayed thus:

```
STAAD.PRO CODE CHECKING - (   AIJ 2002     )
********************************************************************
```

**AD.2007-1003.2.9 IBC 2006**

In **AD.2007-03.1.6 IBC 2006** (on page 217), the graphical screens containing the implementation of the seismic loading chapters of the IBC 2006 and ASCE 7-05 codes were described. In this section, the technical details and command syntax for the parameters of that feature are described.

**Related Links**

- **TR.31.2.12 IBC 2006/2009 Seismic Load Definition** (on page 2592)

**AD.2007-03.2.10 IBC 2006 Response Spectrum**

This command may be used to specify and apply the **RESPONSE SPECTRUM** loading as per the 2006 edition of the IBC specification "International Building Code (IBC)," for dynamic analysis. The graph of frequency–acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

**Related Links**

- **TR.32.10.1.8 Response Spectrum Specification per IBC 2006** (on page 2743)
AD.2007-03.3 Features Affecting the Concrete Design Mode

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2007-03.3.1 ACI 2005 Beam, Column, and Slab Design

The RC Designer mode has been enhanced with a new design module to support the design of beams, columns and slabs to the publication ACI 318-05: Building Code Requirements for structural Concrete and Commentary. This is in addition to the previous ACI design module which is to ACI 318-99.

To create a design brief with the ACI 318-05 design requirements and parameters, on the Groups/Briefs Page start a New Brief by clicking on the New Brief button and select the ACI 318-05 option from the Design Code drop list thus:

![New Design Brief]

**Note:** To perform designs to the ACI 318-05 design code, you will need to have access to a valid license for 'US Design Codes (Standard)'.

AD.2007-1003.3.2 AS 3600 Beam and Column Design

The RC Designer mode has been enhanced with a new design module to support the design of beams and columns to the publication AS 3600-2001: Concrete Structures. The implementation includes both Amendment 1-2002 and Amendment 2-2004.

To create a design brief with the AS 3600 design requirements and parameters, on the Groups/Briefs Page start a New Brief by clicking on the New Brief button and select the AS 3600 option from the Design Code drop list thus:
To design beams and columns with the AS 3600 design brief follow the instructions as described in the RC Designer help (on page 1241).

**Note:** To perform designs to the AS 3600 design code, you will need to have access to a valid license for ‘Canadian/Australian/SA design Codes’.

**AD.2007-1003.3 EC2 2004 Beam and Column Design**

The RC Designer mode has been enhanced with a new design module to support the design of beams and columns to the publication BS EN 1992-1-1:2004. This is the base Eurocode 2 document with the British National Annex.

To create a design brief with the Eurocode 2 design requirements and parameters, on the Groups/Briefs Page start a New Brief by clicking on the New Brief button and select the EC2:2004 option from the Design Code drop list thus:

To design beams and columns with the EC2:2004 design brief follow the instructions as described in the RC Designer help (on page 1286).

**Note:** To perform designs to the EC2:2004 design code, you will need to have access to a valid license for ‘Eurocodes’.

**AD.2007-03.4 Features Affecting the RAM Connection Design Mode**

Several new features have been added and existing features have been modified in the RAM Connection Design Mode. These are explained in the following pages.
Full use of the RAM Connection Mode requires access to a valid RAM Connection license. If you do not possess a license, contact your Bentley account manager to have it added to your SELECT licenses. Without a valid license, only a small subset of the full range of available RAM connections can be utilized.

AD.2007-03.4.1 Support of British Sections

To allow models created with British steel sections to be designed with RAM Connection, these sections need to be added to the supported US section database.

The ranges of supported sections are:

**US Sections**

Wide Flange
- W Shape - W
- M Shape - M
- S Shape - S
- HP Shape - HP

Tee
- Tees cut from W sections - WT
- Tees cut from S sections - ST
- Tees cut from M sections - MT

Channel
- Channel - C
- MC Channel - MC

Angle
- Angle - Equal and unequal, L and LU
  (Double angle, T2L and T2LU, but with fixed spacing)

Hollow
- Tube - TUBE
- Pipe - PIPE
- HSS Rectangular - HSS_RECT
- HSS Round - HSS_RND

Not supported
- B Shape
- Castellated
- Solid Round
- Cable

**British Sections**

Wide Flange
What's New?
STAAD.Pro 2007 Release Reports

UB Shape - UB
UC Shape - UC
JO Shape - Joist

Tee
Tees cut from UB sections - TUB
Tees cut from UC sections - TUC

Channel
Channel - PFC

Angle
Angle - Equal and unequal, EA and UEA

Hollow
Tube - Rectangular and square, SHS and RHS
Pipe - CHS

Not supported
UP Shape

AD.2007-03.4.2 Support of Multiple Connections at a Joint

RAM Connection designs separate parts of a connection with different templates such as a Shear Plate Connection and a Moment Connection. Therefore the RAM Connection mode has been enhanced to allow multiple Connections to be assigned to a single joint.

The following is an enhancement of the new feature added in STAAD.Pro 2007 Build 1001. Refer to AD. 2007-1001.5.1 RAM Connection Design Mode.

If a design brief is assigned to a joint that already has one assigned, then rather than replacing the existing design brief, an additional connection will be defined.
When multiple connections exist at a joint, the information is displayed such that the colour coding of the connection uses the following logic:
At least 1 connection?

yes

All connections have been designed?

no

Cyan

yes

All connections pass?

no

Yellow

More than 1 connection on the joint?

yes

Add No of connections to the joint

Green

When selecting a joint that has multiple connections associated with it, the following dialog box is displayed that shows the associated connections from which the required connection should be selected:

Note: Each connection is designed and detailed independently
The Software Release Report for STAAD.Pro 2007 Builds 01 + 02 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 build 1005. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-1001.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.

AD.2007-1001.1.1 New Start Page

STAAD.Pro now includes a start page with access to the functions normally required when first starting STAAD.Pro, including shortcuts for starting a new file, accessing the recently accessed files, launching the help file and configuring STAAD.Pro.

The new Start Page is divided into 4 sections that can be used to achieve the following:

1. **Project Tasks**, to:
   - a. Start a New Project using the STAAD.Pro wizard.
   - b. Open an existing file using the traditional windows browse dialog enhanced with a model preview window.
   - c. Set the program behavior with the Configuration options.
   - d. Setup the automatic Backup configuration requirements.

2. **Recent Files** to preview and access the last 6 models opened.

3. **Help Topics**, to:
   - a. Quick access to the online Help document.
   - b. Locate the technical support centers and contact details.
c. Find out the latest information on the program online from the Product News link.

4. License Configuration. To quickly identify which SELECT licenses are being used by the current session of STAAD.Pro, they are displayed and color coded on the Start Page thus:

- If the license is available it is marked with a green circle thus:

- Licenses that have not been selected are marked with a grey circle thus:

- If the selected license can not be obtained or is not available from the server will be shown with a red circle, thus:

Additional configuration of the Bentley SELECT license, such as specifying the server name and activation key, can be viewed and set using Bentley License Management Tool which can be accessed from the link at the bottom right of the License Configuration section.

AD.2007-1001.1.2 Enhanced Grid Tool

The Snap/Grid Node tools have been enhanced to

1. Allow multiple different grids to be created.
2. Import a DXF file and use it as a template.
3. Import grid files created in another STAAD.Pro model

Beams, plates (both triangular and quadrilateral) and 8 noded solid elements can be generated using the appropriate Snap/Grid Node tool.

When this function is launched, the following dialog is opened which will include a Default Grid. This grid will be of type ‘Linear’, there are also options to create Radial, Irregular and imported DXF grids which will be described later.
As new grids are added or modified, the information is stored in the STAAD.Pro data folder with a GRD extension that allows other STAAD files to re-use these defined grids.

The effect of the current grid settings are displayed in the graphics window, thus:
To change the settings of this grid click on the **Edit** button to display the current grid properties, e.g:
The current plane of the grid is set by selecting the required option. This can be rotated about one of the global planes by selecting the axis of rotation and setting the angle.

The origin of the grid is marked on the graphics with a small circle:

![Image of grid origin](image)

The location of the origin, specified in global co-ordinates, can either be defined explicitly in the given X, Y and Z co-ordinates, or it can be set to the co-ordinates of an existing node by clicking on the icon,  (the cursor changes to  ) and then on the node itself in the graphical window. Note that at this point the origin co-ordinate is updated.

The construction lines are used to specify how many gridlines are created either side of the origin, the spacing between the gridlines and if there should be a skew in degrees along either axis.

Click on the **OK** button to accept these settings.

Additional grids can be defined by clicking on the [Create...] button. Three different types of standard grid can be created:

- Linear
- Radial
- Irregular

The type of grid required should be selected from the drop list of types available at the top of the property sheet. Each new grid should be identified with a unique name for future reference. The functionality for each type of grid is thus:

**Linear**

The Default Grid defined above is a Linear Grid and thus see above for the settings of a Linear Grid.

**Radial**

The settings for a Radial grid are defined in the following window:
The Plane, Angle of Plane and Grid origin options are as for the Linear (or Default Grid).

The construction lines options

**Start Angle**, is the angle in degrees about the orthogonal axis to the plane from the axis first referred to in the definition of the plane. For example, if the selected plane is X-Y, then the angle is measured about the Z axis (using the right hand rule) from the axis parallel to the X axis.

**Sweep** is the angle in degrees measured from the start angle which is divided into the selected number of Bays, thus:

---

*Irregular*

The settings for an Irregular grid are defined in the following window:
The origin is set as described above for both Regular and Radial grids.
The plane of the grid can either be set in one of the global planes X-Y, X-Z or Y-Z and rotated about one of the
global axes. This method is identical to that described for the Regular or Radial Grids. Alternatively, the
directions of the two axes can be specified as relative co-ordinates from the origin:

The gridlines are defined by the distance to the next gridline and the numbers separated with a space.
There are two import options that can be selected that can allow either DXF files or grids defined in another STAAD.Pro model (all but the default will be imported).

The option to import a DXF file will open the following dialog:

To select a DXF file click on the [...] button and navigate to the required file.

The file will be opened and displayed in the preview window. Individual layers can be turned on and off from the Layers droplist. The individual entities in the selected layers are displayed and can be toggled on or off for import.

By clicking on an entity in the graphical window, the entity is highlighted in the table so that it can be turned off if required.
With the required entities selected, a suitable reference name supplied and unit selected, click on the **Import** button.

The data will be imported in the plane in which it was defined in the original DXF. However, if required this can be rotated about any of the global axes. Also, the origin of the grid can be located at any 3D co-ordinate.

The option to Hide DXF text can be used to toggle the display of grid labels if they start clashing with the rest of the model. The grid is displayed thus (Note curved lines are currently not imported):
The DXF grid operates as the other forms of grid in that when the Snap Node/... button is clicked, nodes can be created at the ends and intersections of grid lines.

The second import option is to Import Grids previously defined in another STAAD.Pro model. Selecting this option opens a browse dialog box to identify a GRD file created by the Snap Node Grid tool. Note that GRD files are only created by STAAD.Pro 2007 (or later).

Icons on the Geometry toolbar are:

- Snap/Node Beam
- Snap/Node Triangular Plate
- Snap/Node Quadrilateral Plate

AD.2007-1001.1.3 Fly-out Toolbars

The amount of screen space occupied by a number of toolbar icons has been recovered by collapsing a number of similar icons into a single icon.

The active icon can be changed by holding down the left mouse button when clicking on the button. Icons that have this property are identified with a black triangle in their lower right corner:

There are four Geometry Toolbar icons that have this property:

- Add Beam
- Add Plate
- Add Solid
What's New?
STAAD.Pro 2007 Release Reports

- Snap Node Grid

The Add Beam icon supports four commands:
  - Add Beam from Node to Node
  - Add Curved Beam
  - Add Beam Between Mid Points
  - Add Beam Using Perpendicular Intersection

The Add Plate icon supports two commands:
  - Add Quadrilateral Plate
  - Add Triangular Plate

The Add Solid icon supports five commands:
  - Add 8 Noded Solid
  - Add 7 Noded Solid
  - Add 6 Noded Solid
  - Add 5 Noded Solid
  - Add 4 Noded Solid

The Snap Node Grid icon supports three icons:
  - Snap Node Beam
  - Snap Node Quadrilateral Plate
  - Snap Node Triangular Plate

AD.2007-1001.1.4 Physical Member Query

The addition of the Physical Member to the range of objects supported by STAAD.Pro requires that there is access to a query tool to get information about the Physical Member.

To access information on a Physical Member, use the Physical Member Selection Tool to highlight the required object and then select the menu option: Tools > Query > Physical Member to highlight the required object and then select the menu option:
This dialog box can also be accessed from

- Double clicking on a physical member with the **Physical Member Cursor**
- Selecting the physical member, right-clicking and selecting **Properties...** from the pop up menu.

**AD.2007-1001.1.5 In-Plane Area Loads on Plates**

STAAD.Pro can now apply plate loads in the local X and Y directions to represent in plane friction loads. As shown in the next figure, pressure on the full element can now be applied along local X and Y axes.

Thus a -2 kip/ft² load applied to an element that is 3 ft² will result in -6 kips being applied in the plane of the element.

This applies only to **Pressure on Full Plate**.

**Related Links**

- **TR.324.1 Area Load Specification** (on page 2665)
AD.2007-1001.1.6 Front-to-Front Channels

STAAD.Pro has been enhanced to support the definition of steel channel sections being defined in an arrangement with the toes pointing to each other with a given spacing.

![Front-to-Front Channel Diagram]

Front to front channel steel sections can be defined in the Properties dialog box by selecting the required channel section and choosing the **FR (Double Channel – Front to Front)** option and any spacing required between the channels.

The command that appears in the STAAD.Pro STD data file is:

```
<Member List> TABLE FR C4X5 SP 0.5
```

**Note:** The SP parameter is optional, but if it is not set, the section will *not* be assumed to be a closed box for torsional calculations.

AD.2007-1001.1.7 Automatic Property Calculations for User-Provided Table Angle Sections

User-Provided Tables (UPTs) of Angle sections require section properties that can now be calculated when the section dimensions have been defined.

Angle Section UPTs require the following data to be entered for each section:
What's New?
STAAD.Pro 2007 Release Reports

1. D, Depth of angle
2. WF, Width of angle
3. TF, Thickness of flanges
4. R, radius of gyration about principal axis, shown as r(Z-Z) in the AISC manual (see below). This must not be set to zero.
5. AY, Shear area along Y axis
6. AZ, Shear area along Z axis

Define the dimensions of the angle (shown in bold above):

Then click on the Calculate button to have STAAD.Pro calculate the remaining properties provided:
**AD.2007-1001.1.8 Consolidation of Multiple Property References**

When a STAAD file has a number of references of the same property, there is now a tool to consolidate all these properties into a single command.

STAAD models that have the same property defined multiple times can be consolidated by clicking on the new menu item, **Tools > Merge Properties**.

A warning is given and thus an opportunity to cancel the property collation if not required or selected by mistake.

All instances of a given section property will be collated into a single property reference.

**Notes:**

- Properties references with differing additional parameters will not be collated.
- Properties references with differing assigned material properties will not be collated.

**AD.2007-1001.1.9 Section Property Reduction in Analysis to Account for Cracking**

Concrete design specifications recommend the use of cracked section properties for the analysis and design of concrete sections. Though the methodology to handle cracked section properties is non-linear in nature, i.e. the section capacities should be checked and modified depending upon the section forces the section is handling. The model should then be re-analyzed with modified reduced section properties and redesigned. This iteration should be continued until the forces in all sections designed are below the allowable limit of ultimate strength.

In STAAD.Pro this approach has been simplified as per the recommendations in the ACI-318: 2005 standard which suggests a user input of reduction factors for the individual members. Section 10.11.1 of ACI-318 has provided a list of suggested reduction factors of section properties dependant upon the nature of stresses the member is subjected to.

An additional sheet has been added to the Beam Specifications dialog to allow the reduction of section properties for analysis to be created and assigned. The method is identical to that for creating any other beam specification.

The Specifications dialog can be accessed from the **General | Specifications Page** in the Modeling Mode:
The reduction factor should be a fraction of unity; hence a factor of 0.5 defined for RAX will reduce a defined cross sectional area of 0.5 ft\(^2\) to 0.25 ft\(^2\).

The format of the command that is generated in the STAAD.Pro STD file is:

```
MEMBER CRACKED
<Member List> REDUCTION { RAX | RIX | RIY | RIZ} factor
```

Multiple factors can be assigned on the same line.

**Note:** The reduction factor is considered only for analysis but not for design.
What's New?
STAAD.Pro 2007 Release Reports

AD.2007-1001.1.10 Tension/Compression-Only Spring Support

STAAD.Pro can now graphically define spring supports that are to allow tension only or compression only forces.

The Tension/Compression Only Support command is created from the Supports dialog box.

The Supports dialog can be accessed from the General | Support page in the Modeling mode.

Click on the Create button and select the Tension/Compression Only Spring sheet thus

The selection of Reaction Type indicates that if, after any of the cycles of analysis, the direction of the force in the spring is of the wrong 'type', then the support will be removed from that direction and a new analysis performed.

<table>
<thead>
<tr>
<th>Reaction Type</th>
<th>Tension Only</th>
<th>Compression Only</th>
</tr>
</thead>
<tbody>
<tr>
<td>Support will remain if the reaction is</td>
<td>- ve</td>
<td>+ ve</td>
</tr>
<tr>
<td>Support will be removed and a new analysis flagged if the</td>
<td>+ ve</td>
<td>- ve</td>
</tr>
<tr>
<td>reaction is</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

What's New?
STAAD.Pro 2007 Release Reports

STAAD.Pro
User Manual
This support definition should be either ‘added’ to the Supports dialog (by clicking on the Add button) or assigned to the currently selected nodes (by clicking on the Assign button) that have previously been assigned with spring supports. If the support is not assigned as it is created, it can be assigned later from the Support dialog. However, note that if it is not assigned to at least one spring support when STAAD.Pro is closed, then the definition will not be saved in the STD file.

**AD.2007-1001.1.11 Enhanced Elastic Mat Plate Mat Options**

The Elastic Mat and Plate Mat commands can be set to behave as compression only springs and include the influence area that each node has been subjected to in the ANL output file.

In order to allow Elastic Mat and Plate Mat commands to perform as compression only supports to model lift of support situations, the command has been enhanced with an additional parameter which can be set graphically from the command definition.

The Elastic Mat and Plate Mat commands are created from the Foundation sheet in the Supports dialog box. The Supports dialog can be accessed from the General | Support page in the Modeling mode.
When the Compression Only option is set, then if after any of the cycles of analysis, the force at a node included in the command range (in the elastic mat range or used to define a plate in the plate mat range) is found to be tensile (i.e., negative reaction), then the load case is marked for a re-analysis with that support removed.

There is also a new option to include in the output file, the area that has been used in the calculation of the spring stiffness of each joint used when defining a Plate Mat or Elastic Mat command.

**Related Links**
- [TR.27.3 Automatic Spring Support Generator for Foundations](on page 2517)

**AD.2007-1001.1.12 Reference Load Cases**

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. This is similar to a REPEAT LOAD command, but has the added benefit of not being solved in its own right.

A reference load case is listed in the Load dialog box of the General Load Page of the modeling mode. A Reference load case is listed in the Definitions section of the data file and displayed thus:
To add a new Reference load case, highlight the Reference Load Definitions in the Load dialog and click on the Add button. Provide the title for the load case and number (however, similar to that of creating Primary load cases, the next available reference load case number. Note that these reference load case numbers can be the same as a primary or combination load case number (however a combination load case number cannot be the same as a primary load case number).

The Reference load case names are shown in the Load dialog box. Load items should then be added to this load case in exactly the same way as adding load items into a primary load case.

Select the Reference load case and click on the Add button to display all the load items that can be added to this load case, thus:
To be analyzed, the loads defined in a Reference load case must be added to one or more Primary load cases. This is done by selecting the required primary load case and clicking on the Add button.

Select the load item **Repeat Load > Reference Load**: 

Select the required defined Reference load cases and click on the [>] button to select them for inclusion in the current load case. The >> button will select all defined Reference Load cases.

These are added with a factor of 1.0, however, each can be modified with its own specific multiplication factor:
The format of the definition of a Reference Load (i) in the data file is thus:

```
DEFINE REFERENCE LOADS
LOAD R(i) LOADTYPE (type) TITLE REF LOAD CASE 1
(LOAD items)
...
END DEFINE REFERENCE LOADS
```

The format of a reference to a Reference Load in a primary load (j) case is thus:

```
LOAD (j) LOADTYPE (type) TITLE LOAD CASE 1
REFERENCE LOAD R(i) 1.0
...
```

**Related Links**
- *Repeat Load tab* (on page 3028)
- *TR.31.6 Defining Reference Load Types* (on page 2642)
- *M. To create a reference load* (on page 873)

**AD.2007-1001.1.13 Enhanced Beta Angle Definition and Assignment**

Assigning beta angles on members has been improved visually by displaying the commands that are stored in the STAAD data file, making them easier to manage.

Beta angle definitions are created and listed in the Beta Angle sheet of the Property dialog box. Each new command is listed in the dialog as shown below and can be assigned using the assignment methods commonly used throughout STAAD.Pro:
The creation of beta angle commands includes three options:

1. Define specific angle to rotate the beams about their local X axis.
2. Specify the command ‘Angle’ (*)
3. Specify the command ‘RAngle’ (*)

(*) The Angle and RAngle commands are specifically for equal and unequal angle sections. All sections are aligned with their principal axes aligned with the global axes, however, angle sections are often required to align their flanges with the global axes. By assigning either the Angle or RAngle command, the section will be rotated to align the flanges. For more information, see section 5.26.2 - Specifying CONSTANTS for members, plate elements and solid elements - of the Technical Reference Manual (on page 2503).

Related Links

The static equivalent method for performing dynamic analysis per the IBC 2006 code has been implemented in STAAD.Pro 2007 Build 02.

This option can be accessed from the General | Load page as explained below.

When the General | Load Page is selected, the right hand side of the screen will display the following if no load cases exist in the model.

Definitions contains the options through which one creates the “Define” block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases.

When the tree view is expanded, it will look as shown below.

Select Seismic Definitions and click on Add.

In the dialog box that comes up, select IBC 2006 from the drop-down list.
In this dialog box, we can specify the various parameters as described below.

**Include Accidental Torsion**
Check this box to calculate the accidental torsion component described in section 12.8.4.2 of ASCE 7-05.

**Parameter**
The various parameters for the IBC 2006 code, such as the Occupancy Importance factor IE, Response modification factors RX and RZ, spectral response accelerations SDS, SD1 and S1, etc., are described in detail in TR.31.2.12 IBC 2006/2009 Seismic Load Definition (on page 2592).

After specifying the values for the parameters, click on the Add button.
We will see that the Load dialog box has now been updated.
Next, we should define the structural weights for calculating Base Shear.

After highlighting the expression LAT 38.0165……, click on Add. A new dialog box titled Add New Seismic Definitions will come up.

Related Links
• M. To add a seismic load definition (on page 840)

This tab allows the user to apply response spectrum loads on the structure.
This option can also be accessed from the General | Load page.
Period vs. Acceleration

Provide the values of period (seconds) and corresponding acceleration (current length units/sec^2) or displacement (current length unit). Spectrum pairs should be provided in ascending value of period. As we provide the curve points, the program displays the curve at the bottom of the dialog box.

Related Links

- **TR.32.10.1.8 Response Spectrum Specification per IBC 2006** (on page 2743)
- **M. To add an IBC 2006 response spectrum** (on page 848)

AD.2007-1001.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.
AD.2007-1001.2.1 P-Delta Analysis Including Stiffening Effect of the KG Matrix

The P-Delta analysis capability has been enhanced with the option of including the stress stiffening effect of the Kg matrix into the member / plate stiffness.

A regular STAAD P-Delta Analysis performs a first order linear analysis and obtains a set of joint forces from member/plates based on the large P-Delta effect. These forces are added to the original load vector. A second analysis is then performed on this updated load vector (5 to 10 iterations will usually be sufficient).

In the new P-Delta KG Analysis, that is, with the Kg option selected, the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is then performed using the original load vector. Large & small P-Delta effects are always included (1 or 2 iterations will usually be sufficient).

The KG option is activated by selecting the option on the P-Delta Analysis dialog thus:

Related Links
- [A. To specify a P-Delta analysis](on page 928)

AD.2007-1001.2.2 P-Delta Analysis Including Small Delta

A regular STAAD P-Delta Analysis can now account for the small P-Delta effect whilst performing a P-Delta analysis.

Without the Small Delta option, i.e. a regular STAAD P-Delta analysis, STAAD performs a first order linear analysis and obtains a set of joint forces, from members/plates based on the large P-Delta effect, which are then added to the original load vector. A second analysis is then performed on this updated load vector.
With the Small Delta option selected, both the large & small P-Delta effects are included in calculating the end forces, (5 to 10 iterations will usually be sufficient).

The option is activated by selecting the option on the P-Delta Analysis dialog thus:

Related Links
•  *A. To specify a P-Delta analysis* (on page 928)

### AD.2007-1001.2.3 Buckling Load Analysis

STAAD.Pro can now identify the factor by which the loads in the selected load case should be increased (or decreased if less than 1) such that Euler buckling would occur.

Two methods have been introduced to do buckling analysis. One method is introduced in the standard solver, the other in the advanced solver, as described below.

**Related Links**

•  *A. To specify buckling analysis* (on page 932)

### AD.2007-1001.2.3.1 Buckling Analysis Using the Basic Solver

By including the command PERFORM BUCKLING ANALYSIS, the program will perform a P-Delta analysis including Kg Stiffening (geometric stiffness of members & plates) due to large & small P-Delta effects. If a non-singular stiffness matrix can be created, then buckling has not occurred. Then the load is increased from the last increment repeatedly until buckling does occur. Then the load is decreased halfway back to the prior increment. This bounds the buckling factor between the last 2 increments. Then STAAD proceeds to halve the interval until either the change between increments is 0.1% of each other, or the specified number of increments has been exceeded. The resulting factor is reported in the output file. The buckling deformed shape is simply the deformed shape from a static analysis with the near buckling load applied. This could appear more like a crushing, small displacement shape rather than a buckling mode shape. 15+ iterations are recommended.

Buckling will be applied to all primary cases.

The option is activated using the new option in the **Analysis/Print** dialog thus:
The results of the Buckling analysis are presented in the output file thus:

AD.2007-1001.2.3.2 Buckling Analysis Using the Advanced Solver

This buckling method is automatically activated if an Advanced Analysis license is available. When using the Advanced Solver, the corresponding ‘buckling modes’ are included in the output file. [See AD.2007-1001.2.6 Advanced Solver (on page 267) on the procedure to invoke the alternative standard buckling load analysis solver the use of which is explained in section AD.2007-1001.2.3.1 (on page 263) of this document]

The option is activated using the new option in the Analysis/Print dialog thus:
The program performs a P-Delta analysis including Kg Stiffening (geometric stiffness of members & plates) due to large & small P-Delta effects.

The eigensolution,

$$| [K] - BF [Kg] | = 0$$

is solved for the buckling factors and buckled mode shapes. The first 4 buckling factors and buckled shapes are calculated and included in the output file:

The buckling modes and shapes are available to be viewed in the Post Processing Mode in a new Buckling Page.
This page includes both a Buckling Factors table:

<table>
<thead>
<tr>
<th>Mode</th>
<th>Buckling Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.426</td>
</tr>
<tr>
<td>2</td>
<td>2.855</td>
</tr>
<tr>
<td>3</td>
<td>4.747</td>
</tr>
<tr>
<td>4</td>
<td>4.802</td>
</tr>
</tbody>
</table>

and a Buckling Modes table:

Only the primary load case just prior to the PERFORM BUCKLING command is used. The number of iterations entered is ignored. The buckling factor result is reported in the output file and in post processing.

**AD.2007-1001.2.4 Modal Analysis Including Stress Stiffening Effect of KG Matrix**

STAAD.Pro can include the stress stiffening effect (geometric stiffness) based on the axial member forces/plate in-plane stresses from a selected load case when calculating the modes & frequencies of a structure.

Position the selected load case from which the axial stresses are to be used to modify the stiffness matrix, such that it is the last static case before the dynamic case which is in turn immediately followed by a PDELT A KG command.

The dynamic load case should contain mass data followed by one of the following:
a. A MODAL CALCULATION REQUESTED command.
b. A response spectrum definition, i.e., set of SPECTRUM command data.
c. A reference to a time history definition, i.e., include TIME LOAD commands.
d. Valid Steady State data.

Example

...  
LOAD 1 LOADTYPE None TITLE LOAD CASE 1  
******* This is static loading case from which the “axial stress” is  
******* used to compute the stress stiffening effects (P-Delta)  
******* This case will be solved as a PDelta case with large & small  
******* P-Delta effects  
SELFWEIGHT Y -1.0  
JOINT LOAD  
2 3 6 7 9 TO 12 FY -3  
LOAD 2 LOADTYPE None TITLE LOAD CASE 2  
******* Enter masses in weight units  
SELFWEIGHT X 1  
SELFWEIGHT Y 1  
SELFWEIGHT Z 1  
JOINT LOAD  
2 3 6 7 9 TO 12 FX 10 FY 10 FZ 10  
******* Declare this to be a modes/freq analysis  
******* Note that dynamic cases use the factored matrix from the last *******  
load case; which is a (K+Kg) case  
MODAL CALCULATION REQUESTED  
PDELTA KG ANALYSIS

AD.2007-1001.2.5 Enhanced Master/Slave Command Processing

The internal processing of any Master Slave command has been enhanced to allow an automatic bandwidth reduction to take place.

The analysis engine now performs a bandwidth reduction on files that include Master/Slave commands which must occur in the input file after the definition of supports. In previous versions of STAAD.Pro, for the bandwidth reduction to take place, the data of Master/Slave would need to be repeated before the support definitions. This requirement is now no longer required.

Related Links
• TR.28.1 Master/Slave Specification (on page 2525)

AD.2007-1001.2.6 Advanced Solver

A new substantially faster analysis engine has been produced which can provide solutions of large structures in a fraction of the time currently required by the standard STAAD engine. The Advanced Solver generally uses less disk and memory as well.

The Advanced solver is a new addition to the STAAD Analysis Engine which can be used for solving both static and dynamic problems. It is part of the STAAD engine with no special command required to run it. It is automatically activated if a suitable license is available (*), however, this can be turned off and the standard solver used by including the option:

SET STAR 0
This command must be included in the header information block at the start of the file and before the first JOINT command block.

The engine can operate in two modes, *in-core* and *out-of-core*. The in-core solver will be used for models with under 20000 joints and the out-of-core solver for models over 20000 joints. In most situations, the in-core mode will provide the quickest solution, but where there is insufficient memory available, then the engine will use the out-of-core mode. Again, selection of the mode is automatically chosen by the analysis, but can be over-ridden.

The full set of overrides for the advanced engine is:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET STAR -3</td>
<td>use in-core solver regardless of size</td>
</tr>
<tr>
<td>SET STAR 4</td>
<td>use out-of-core solver regardless of size</td>
</tr>
<tr>
<td>SET STAR 3</td>
<td>default</td>
</tr>
<tr>
<td>SET STAR 0</td>
<td>use STAAD standard solver</td>
</tr>
</tbody>
</table>

**Notes:**

(*) To use this feature requires access to a ‘STAAD Advanced’ license. If you do not currently have this feature, please contact your account manager.

(**) Global Euler Buckling analysis is different between the two solvers. See AD.2007-1001.2.3 Buckling Load Analysis (on page 263) for the differences.

**AD.2007-1001.2.7 Eurocode 3:2005**

The latest UK release of *Eurocode 3: Design of steel structures – part 1-1: General Rules and rules for buildings* has been implemented.

This code is an update to our current Eurocode 3. The changes are minor and the code checking process remains the same as the previous Eurocode 3 design.

The design parameters are the same and apply as defined for the current Eurocode 3, the only exception, however, is the ‘Code’ parameter which must be set to ‘EC3 BS’. This setting is available form the GUI list of codes.

Note that a design performed to the new Eurocode 3 standard is displayed in the output file (*.ANL) with the following header:

```
STAAD.PRO CODE CHECKING - (BS EN 1993-1-1:2005)
**************************
PROGRAM CODE REVISION V1.1 BS_EC3_2005/1
```

The equivalent header for a code check (or member selection) to the older standard is displayed thus:

```
STAAD.PRO CODE CHECKING - (DD ENV)
**************************
PROGRAM CODE REVISION V1.14_EC3_94/1
```

**What's New?**

STAAD.Pro 2007 Release Reports

In AD.2007-1001.1.14 IBC 2006 Equivalent Lateral Force Procedure (on page 258), the graphical screens containing the implementation of the seismic loading chapters of the IBC 2006 and ASCE 7-05 codes were presented.

Related Links
- M. To add a seismic load definition (on page 840)


This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2006 edition of the ICC specification "International Building Code (IBC)," for dynamic analysis. The graph of frequency–acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

Related Links
- TR.32.10.1.8 Response Spectrum Specification per IBC 2006 (on page 2743)
- M. To add an IBC 2006 response spectrum (on page 848)

AD.2007-1001.3 Features Affecting the Post-Processor (Results Mode)

A new feature has been added to the post-processor section of the program, also known as the Results Mode. It is explained in the following pages.

AD.2007-1001.3.1 Statics Check Table

The equilibrium check that has been available in the output file is now automatically included in the Post Processing Mode.

The Node | Reactions Page has been enhanced with a new table that displays the results of an equilibrium check. This is the same information that in the past would have been only available by including a PRINT STATICS CHECK in the perform analysis command.

The Statics Check Results table can be accessed from the Nodes | Reactions page in the Post-Processing mode.

AD.2007-1001.4 Features Affecting the Concrete Design Mode

The enhancement made in the RC Designer section of the program, also known as the Concrete Mode, is explained in the following pages.
AD.2007-1001.4.1 Beam and Column Designs to the Russian Concrete Code SP52

Two new design options have been added that allow concrete beams and columns to be designed to the Russian Concrete design code SP52-101-3.

The Russian Design Codes are only available with the **Euro-zone – Eastern Design Codes** SELECT license. The SP52-101-03 design checks are initiated by creating a design brief based on this code. This option is now available and can be selected when creating a new design brief. Beams and columns can be designed using design briefs tailored to the requirements of the specific item.

Design Briefs are created in the **Design Layer | Groups/Briefs** Page of the Concrete Designer mode. Click on the **New Brief** button on the base of the Design Brief Table:

![New Brief button](image)

**Beam Design Briefs**

An SP52 Beam design brief is created by setting the design brief options in the dialog that is presented when the **New Brief** button is clicked on the Design Briefs dialog:

![Beam Design Brief dialog](image)

The beam design brief contains two sheets of data that are used in the calculation of the reinforcement of any member that it is associated with. Default values are provided which should be changed to the specific values required for the design. These sheets are titled ‘General’ and ‘Reinforcement’ thus:
Column Design Briefs

An SP52 Column design brief is created by setting the design brief options in the dialog that is presented when the New Brief button is clicked on the Design Briefs dialog:
The column design brief contains three sheets of data that are used in the calculation of the reinforcement of any member that it is associated with. Default values are provided which should be changed to the specific values required for the design. These sheets are titled 'General', 'Reinforcement' and 'Member Loadcases' thus:
With the Design Brief created, Design Groups must be defined. These are created by selecting the design members that are required and clicking on the **New Design Group** button on the base of the Design Groups dialog:
What's New?

STAAD.Pro 2007 Release Reports

This pops up a dialog to allow the required Design Brief to be assigned to the selected design members that are now in this Design Group.

The group is added to the Design Group table and by double clicking on the group label, e.g. G1, then the list of members in the group is displayed and can be edited.

Note from here on, a suitable license is required to perform a design or review the results of a design.

To perform a design or review results of a previous design, requires entering the Concrete Member Mode by either clicking on the tab on the left of the screen or selecting Member Design from the Mode menu.

The program uses the Design Brief of the current Design Group to determine the page tabs and page layout that is now presented. The current Design Group is selected by choosing it from the drop-list of design groups displayed at the top of the screen.

*Beam Designs*

When the current group has a beam design group associated with it, entering the Concrete Member Mode, there are the following 7 pages:

- Summary
- Design
- Main Layout
- Main Reinforcement
- Shear Layout
What's New?
STAAD.Pro 2007 Release Reports

- Shear Reinforcement
- Cracks

The **Summary** Page displays the current status of the design of the members in the design group in a Summary table. Each member is identified with the principal checks including the span and section where the maximum utilization occurs. If any section cannot achieve a solution such that the design ratio is greater than unity, this is identified in red, thus:

![Summary Page example](image)

The **Design Page** is laid out in the same manner as the Summary Page but also launches the Design options dialog, where members from the design group can be selected or removed from the design process:

![Design Options dialog](image)

The **Main Layout** Page displays the design reinforcement for the current member selected from the design group drop list:
Note that the section arrow displayed on the elevation of the beam or the moment diagram, can be moved to display the reinforcement at any location along the beam.

By clicking on either the beam elevation or the cross section, then clicking on the 'Take Picture' toolbar icon, a picture will be added to the picture album which can be included in a User Report:

The **Main Reinforcement** Page displays the summary of the calculations at each of the design sections for both the maximum positive and negative moment at the given section for the current member.

The **Shear Layout** Page displays the shear reinforcement for the current member:
The Shear Reinforcement Page displays the summary of the shear reinforcement calculations at each cross section for the current member.

The Cracks Page displays the summary of the crack width check for both short and long terms and both maximum hogging and sagging moments:

Column Design

When the current group has a column design group associated with it, entering the Concrete Member Mode, there are the following 6 pages:

- Summary
- Input
- Design
- Main Layout
- Results
- Drawing

The Summary Page displays the current status of the design of the members in the design group in a Summary table. Each member is identified with the bars provided and a safety margin, thus:
The **Input** Page provides a table to set the effective length factors (enter b to reset to the value set in the brief), an indication as to whether either direction is braced and what size link bar is to be used thus:

The **Design** Page is laid out in the same manner as the Summary Page but also launches the Design options dialog, where members from the design group can be selected or removed from the design process:

The **Main Layout** Page displays the reinforcement layout in the column, including the shear links. For a typical rectangular or square column it is shown thus:
For a circular column it is shown thus:

The Results Page displays a table of the principal check for each column in the group thus:

The Drawing Page allows the current column details to be exported into a DXF file that can be used imported into a CAD package. Also a simple summary schedule of bars for the current member is displayed.
Printing Reports

Refer to the RC Designer manual (on page 1016) for more information on creating user reports.

Exporting to Bentley Rebar

Both beam and column designs can be exported to Bentley Rebar by selecting the menu option, Groups > Export Group to Rebar.

For more information, see ‘What’s New in STAAD.Pro 2006 Build 1004’ AD.2006-1004.4.1 Bentley Rebar Export from RC Designer.

AD.2007-1001.5 Features Affecting the Connection Design Mode

A new mode has been added that allows quick design of connections using the RAM Connection application. This is displayed on the Modes toolbar with the above icon.

Full use of the RAM Connection Mode requires access to a valid RAM Connection license. If you do not possess a license, contact your Bentley account manager to have it added to your SELECT licenses. Without a valid license, only a small subset of the full range of available RAM connections can be utilized.

AD.2007-1001.5.1 RAM Connection Design Mode

There is now a new mode in STAAD.Pro to dynamically link structural model data, including section properties and analysis results, to the RAM Connection application to check connection designs for code compliance. The resulting data and diagrams of the connection can also be included in the User Report.

Connections are designed in the RAM Connection Mode by creating ‘Joints’, from the geometry, section properties and forces resulting from the analysis and assigning a ‘Design Brief’ of connection templates from which a suitable connection, where available, is reported.
Refer to the D. Connection Design workflow (on page 953) for current details on this feature.

AD.2007-1001.6 Features Affecting the Piping Mode

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Piping Mode. These are explained in the following pages.

AD.2007-1001.6.1 Persistency of Pipe Models

When a pipe model is loaded in the Piping Mode, the location of the data is retained along with the connections defined in the Supports Page. Thus, if STAAD.Pro is closed and re-opened, it is no longer necessary to redefine the pipe model or the support relationships.

This feature is an enhancement of the functionality of the Piping Mode.

STAAD.Pro 2006 Release Reports

The Software Release Report for STAAD.Pro 2006 contains detailed information on additions and changes that have been implemented since the final build of STAAD.Pro 2005. This document should be read in conjunction with all other STAAD.Pro manuals.

STAAD.Pro 2006 Build 1004 Release Report

The Software Release Report for STAAD.Pro 2006 Build 1004 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 build 1002. This document should be read in conjunction with all other STAAD.Pro help.

AD.2006-1004.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modelling Mode. These are explained in the following pages.

AD.2006-1004.1.1 New AISC Unified Code

A new steel design code module has been added to STAAD.Pro, which allows steel beam sections to be designed to the rules defined in AISC specification, Specification for Steel Buildings, March 9 2005, ANSI/AISC 360-05. This is also referred to as the AISC Unified Code. The GUI has been updated to allow you to enter the design information for this new design code.

To perform a design of steel sections to the AISC Unified Code, a design parameter block containing the command CODE AISC UNIFIED followed by the command to perform a CODE CHECK or MEMBER SELECTION must be added to the STAAD.Pro data file.

The GUI has been updated by adding AISC UNIFIED in the Steel Design dialog:
When this design code option is selected, clicking on the Define Parameters button at the base of the dialog box will in turn open the Design Parameters dialog that will allow parameters to be added or added and assigned to specific steel members.

**Note:** Design to this module is only possible if your security allows design to US design codes.

**Related Links**
- *D1.A. American Codes - Steel Design per AISC 360 Unified Specification* (on page 1366)
- *AD.2006-1004.1.2 Update Indian IS 456 Concrete Code*

Two new parameters have been added to the IS 456 design code to assist in the design of concrete columns. ULY and ULZ are used to define the ratio of unsupported to actual column length about the local Y and Z axes.

To include the design parameters for a IS456 design check, click on the Design, Concrete Page, and select IS456 from the drop-down list in the concrete design dialog.

When this design code option is selected, clicking on the Define Parameters button at the base of the dialog box will in turn open the Design Parameters dialog that will allow parameters to be added or added and assigned to specific steel members.
The definition of the parameters and their default values are as defined below:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ULY</td>
<td>1.0</td>
<td>Ratio of unsupported length to actual length of column about minor axis.</td>
</tr>
<tr>
<td>ULZ</td>
<td>1.0</td>
<td>Ratio of unsupported length to actual length of column about major axis.</td>
</tr>
</tbody>
</table>

**AD.2006-1004.1.3 New NRC 2005 Seismic Code**

A new seismic definition module has been added to the STAAD.Pro Loading Definitions. This is to provide the user a tool to calculate and distribute the equivalent static loads produced by an earthquake that should be considered in a building as defined in the ‘National Building Code of Canada 2005 – Volume 1’, Section 4.1.8. This is also known as the NRC 2005 Seismic provisions.

This option has been added to be in addition to the previously available 1995 version of the code.

The NRC 2005 seismic provisions can be added using the **Seismic Definitions** dialog from the Definitions option in the main Load dialog box and selecting **CANADIAN NRC – 2005** from the type pull-down list.
Once all the required values for the parameters have been defined, then clicking on the Add button will add the data to the model such that it is ready to be used in seismic load cases.

For full details of the command structure for the NRC 2005 parameters see Section 5.31.2.11 of the Technical Reference manual (on page 2552).

Related Links
- TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1 (on page 2552)
- AD.2006-1004.1.4 New Turkish Seismic Code

A new seismic definition module has been added to the STAAD.Pro Loading Definitions. This is to provide the user a tool to calculate and distribute the equivalent static loads produced by an earthquake that should be considered in a building as defined in the ‘Specifications for Structures to be Built in Disaster Areas. Part III – Earthquake Disaster Prevention. Amended on 2.7.1998, Official Gazette No. 23390 (English Translation)’. This is referred to as the Turkish Seismic Provisions.

The Turkish seismic provisions can be added using the Seismic Definitions dialog from the Definitions option in the main Load dialog box and selecting TURKISH from the type pull-down list.
Once all the required values for the parameters have been defined, then clicking on the Add button will add the data to the model such that it is ready to be used in seismic load cases.

For full details of the Turkish seismic load command structure see Section 5.31.2.12 of the Technical Reference manual (on page 2613).

Related Links
• TR.31.2.19 Turkish Seismic Code (on page 2613)

AD.2006-1004.1.5 Update Japanese Seismic Definition

The current version of the Japanese Seismic Loading Definition has been enhanced to allow the user to specify a value of ‘α (alpha)’, which is the ratio of frame height to the overall building height, which previously been set to 0.8 and not possible for the user to change.

The Japanese seismic provisions can be added using the Seismic Definitions dialog from the Definitions option in the main Load dialog box and selecting Japanese (AIJ) from the Type pull-down list.

Once all the required values for the parameters have been defined, then clicking on the Add button will add the data to the model such that it is ready to be used in seismic load cases.

Note: The implementation of the Japanese Seismic Definition is as per Article 88 in the ‘Building Codes Enforcement Ordinance 2006.’

For full details of the Japanese seismic load command structure see Section 5.31.2.4 of the Technical Reference manual (on page 2605).

Related Links
• TR.31.2.16 Japanese Seismic Load (on page 2605)

AD.2006-1004.1.6 New Eurocode 8 Response Spectrum


The provisions for Eurocode 8-2004 have been added to the Response Spectrum dialog box.
Enter the General, Loads Page of the Modelling Mode, and select the load case in the Load dialog box. Select the required load case name from the Load Case Details group and click on the Add button.

In the dialog box that is displayed, select the Response Spectra option from the list of items on the left and select the code Euro (EC8)-2004 to display the following:

![Response Spectrum dialog box](image)

Select the required parameters such as the Load Type and Combination Method and the direction that the spectrum is to be defined and click on the Add button to add the command to the input file.

**Note:** This is response spectrum definition has been provided in addition to the ENV 1998-1-1:1994 version of the code that was previously implemented. The older version of the code is still available, although recommended for comparison purposes only.

For a full description of both implemented versions of the Eurocode 8 Response Spectrum command structure, see Section 5.32.10.1.4 of the Technical Reference manual (on page 2715).

**Related Links**
- TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2715)
- M. To add an EC8 response spectrum (on page 849)

**AD.2006-1004.1.7 Update IS 1893 Dialog to Support Command Structure**

The Response Spectrum dialog for defining a Response Spectrum to be calculated as per the IS-1893 code by the analysis engine has been enhanced so that it reflects the input requirements of this command.

Enter the General | Loads Page of the Modelling Mode and select the load case in the Load dialog box. Select the required load case name from the Load Case Details group and click on the Add button.

In the dialog box that is displayed, select the Response Spectra option from the list of items on the left and select the code IS-1893. There are essentially 2 methods that can be employed to define the response spectrum:

1. Select a SubSoil Class
2. Define a Custom Reponse Graph
This choice is made from the **SubSoil Class** in the Spectrum Table. If a defined class is selected, then the Response Spectrum Graph is automatically defined and specified during the analysis, however the following dialog is presented to allow other options to be set:

![Response Spectrum Graph](image)

However, if a custom Response Spectrum Graph is required and selected from the SubSoil drop list, then the various points on the graph of period/acceleration or period/displacement must be entered, thus:

![Custom Response Spectrum Graph](image)

For a full description of the scope of the IS-1893 Response Spectrum command, refer to [Section 5.32.10.1.1 of the Technical Reference manual](on page 2721).

**Related Links**
- **TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002** (on page 2721)
- **M. To add an IS 1893 response spectrum** (on page 846)
AD.2006-1004.1.8 Preview of Surface Element Meshes

Models that contain large numbers of surfaces, or have complex geometrical requirements can result in extensive analyses which may be unnecessary if appropriate settings are applied to the surface. In order to the mesh that will be generated during the analysis, the option to ‘Preview’ the surface meshes has been added.

After creating one or more surface elements, go to the Geometry, Surface Page which will display the Surface dialog. By clicking on the Preview button at the base of the dialog, the mesh that will be generated by the defined parameters will be displayed thus:
AD.2006-1004.1.9 Key File Option Added to Configuration Settings

The output file has the option to list all the key files that were used with the analysis including the date of the principal files. However, this can produce unwanted extra printout data and thus an option to specify whether or not this is to be used has been set in the Configuration Settings.

To access the File Configuration Settings, start STAAD.Pro, but close any open file or the New Wizard if it is open. Click on the menu option, File > Configure to open the Configuration Settings dialog and click on the sheet Input/Output File Format, thus:
To include the key file information in the output file, set the option **Add key file information to output**. If the option is set, the output file contains information similar to the following at the end of the file.

```
EXAMPLE PROBLEM NO. 1 -- PAGE NO. 18

Information about the key files in the current distribution

<table>
<thead>
<tr>
<th>Modification Date</th>
<th>CRC</th>
<th>Size (Bytes)</th>
<th>File Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>10/26/2006</td>
<td>0x3cc1</td>
<td>13266944</td>
<td>SProStaad.exe</td>
</tr>
<tr>
<td>10/25/2006</td>
<td>0x1cc0</td>
<td>05836800</td>
<td>SProStaadStl.exe</td>
</tr>
<tr>
<td>09/19/2003</td>
<td>0x2fc0</td>
<td>00081970</td>
<td>CMesh.dll</td>
</tr>
<tr>
<td>10/25/2006</td>
<td>0x4601</td>
<td>02486272</td>
<td>dbSectionInterface.dll</td>
</tr>
</tbody>
</table>
```

**Note:** If the general case is that the Key Information is displayed, it can be tuned off in a particular analysis if the data file contains the command `SET NOFILE`. 
AD.2006-1004.1.10 Load Icons Display Option

Certain models have large numbers of loads defined in any given load case. It is now possible to select which 'types' of load are displayed when the load case data is displayed.

To select which load items are displayed when the load is displayed, select the Options from the menu item View > Options and select Load Icons thus:

To return to the default of displaying all load types, click on the button Reset.

AD.2006-1004.1.11 Update Oneway Loading

The ONEWAY command has been enhanced with the option to define the direction in which the load is spanning by selecting a member in the loaded zone onto which the load is to be directed.

Enter the General | Loading Page and create or select the required load case.

Click on the Add button on the Load dialog box.

From the options on the left, select the Floor Load option and enter the Floor load in the normal way defining the direction, Load intensity and range. This defines a two way spanning system, to indicate a one-way span, click on the One Way Distribution option. This will then span in the shorter direction. However, now a beam number can be selected from the Towards drop list which is populate with the beams that exist in the defined range. As shown blow:
The selected beam is highlighted on the structure so that the spanning direction can be confirmed, thus:

Once the command is created, the load and contributing areas can be displayed by highlighting the command in the Load dialog box, thus:
By changing the beam number that defines the direction that the load is towards, the change in the loading can be viewed, thus:

AD.2006-1004.1.12 Additional Standard European Steel Section Types

Additional steel section shapes produced by European Steel manufactures have been added to the standard steel database libraries. Section shapes, HD, HL, UPE have been added.

Click on the General | Property page and on the Properties dialog box, click on the Section Database button to view steel section databases available. Click on the European label to view the section types that are now available.
AD.2006-1004.2 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2006-1004.2.1 AISC Unified Code

A new steel design code module has been added to STAAD.Pro, which allows steel beam sections to be designed to the rules defined in AISC specification, *Specification for Steel Buildings*, March 9 2005, ANSI/AISC 360-05. This is also referred to as the AISC Unified Code. The GUI has been updated to allow you to enter the design information for this new design code.

**Related Links**
- [D1.A. American Codes - Steel Design per AISC 360 Unified Specification](on page 1366)

AD.2006-1004.2.2 ACI 2005

The ACI concrete design module has been updated to support the current 2005 version of the ACI code, ACI 318-05.

When performing the concrete design using the parameter CODE ACI 2005, the design will be performed according to the requirements of the American Concrete Institute document, ‘Building Code Requirements for Structural Concrete (ACI 318-05)’. If it is required to perform the design to the older versions of the code, specifically, the 1999 or 2002 version of the code, then the commands:

```
CODE ACI 1999
```

or

```
CODE ACI 2002
```

The commands CODE ACI or CODE ACI 2005 will set the program ready to perform the design to ACI 2005 when it encounters the commands DESIGN BEAM..., DESIGN COLUMN..., or DESIGN ELEMENT...
For more information on the refer to the updated Section 3 of the Technical Reference manual, American Concrete Design (on page 1478).

AD.2006-1004.2.3 Enhancement of Indian Concrete Design

Two new parameters have been added to the Indian concrete design code, IS 456.

ULY and ULZ which are the unsupported lengths in the local Y and Z directions. These commands have been added to assist in the calculation of the minimum eccentricities.

Related Links
- D8.A. Indian Codes - Concrete Design per IS 456 (on page 1887)

AD.2006-1004.3 Features Affecting the Post Processing (Results) Mode

Several new features have been added and existing features have been modified in the Steel Designer section of the program, also known as the Steel Designer Mode. These are explained in the following pages.

AD.2006-1004.3.1 Unity Check Page Update

The Design Results table has been updated so that not only can the maximum utilisation ratio be displayed, but also the governing clause and combination. The superfluous section dimension information has been removed. Additionally, the table can be sorted by design ratio.

Perform an analysis and design and enter the Post Processing Mode. To view the Design Results table, click on the Beam | Unity Check Page. The table is displayed on the right of the window:

To sort the table by ratio, click on the Ratio column heading. To return to the un-sorted list, click on the Beam column heading.

AD.2006-1004.3.2 View Value Annotation Enhanced

The annotation of forces and displacements has been enhanced to allow the number of decimal places viewed on the diagrams to be configured.
1. Open a model with analysis results or run an analysis to obtain a set of valid analysis results.
2. Enter the Post Processing mode and click on the Beam | Forces Page to display the bending moment on the graphic view of the structure.
3. Click on the menu item Results > View Value... to annotate the bending moment diagram and define the items that are required to be annotated in the Ranges sheet, the list of items to be annotated such as Maximum Bending Moments.

4. To change the configuration,
AD.2006-1004.4 Features Affecting the Concrete Design Mode

Several new features have been added and existing features have been modified in the RC Designer section of the program, also known as the Concrete Design Mode. These are explained in the following pages.

AD.2006-1004.4.1 Bentley Rebar Export from RC Designer

When a design has successfully been carried out, the beams or columns may now be exported to Bentley Rebar from the **Groups > Export Group To Rebar** menu item. This will write the member outline and designed bars into a file for transfer into Bentley Rebar so that any modifications that a detailer feels are appropriate may be made. Bentley Rebar is a much more powerful detailing environment and can produce customised drawings and reinforcement schedules.

1. Open a suitable concrete model (such as Examp08.std) and perform an analysis if there are no analysis results available.
2. Select the Concrete Design Mode and create design members, a Design Group with a suitable Design Brief. Full details on the setup process can be found in the **RC Designer Manual, section 2 Getting Started** (on page 1007).
3. Enter the Concrete Member Mode of the RC Designer and perform a design.
   Note that members that can be successfully designed are color coded green, thus
4. Any of the members that produced a satisfactory design can be exported into a neutral file by clicking on the menu item: **Groups > Export Group to Rebar**.

All the valid designed members are displayed and those that are wanted for detailing should be listed in the 'Selected' column. Those that are not wanted should be moved into the 'Available' column. Once the list is correct, click on the [Export] button to create a 'Bentley Rebar Link File that has the file extension; .BRL
5. Start Bentley Rebar and open a suitable DGN file.
6. From the run the menu command: **Applications > GEOPAK REBAR > Menu**.
   This adds a new Rebar menu.
7. To import the RC Designer Group, click on the menu command: **Rebar > Tools > STAAD Detailing**.
   This opens the STAAD manager dialog from which Templates can be defined to describe how members that
   are imported are defined:

   ![STAAD Manager dialog](image1)

   and allows link files to be imported and located on the DGN file:

   ![STAAD Manager dialog](image2)
What's New?
STAAD.Pro 2006 Release Reports

Note: MicroStation and Bentley Rebar are separate products from STAAD.Pro and are independently licensed. If you do not have copies of these products, please contact your STAAD.Pro provider.

AD.2006-1004.4.2 Addition of Amendments 1,2, and 3 to the BS 8110 Modules

Due to the withdrawal of BS 5328 and its replacement by BS 8500, the terminology for concrete has changed from “grade” to “strength class”, and this is reflected in the design briefs. Also, due to harmonisation with European steel codes, Grade 500 high yield steel is now the standard, instead of Grade 460. The main changes in the British Standard were to change the partial safety factor for steel to 1.15 instead of 1.05. To allow older designs to be checked, the program now allows you to specify both the steel strength and the partial safety factor to be used.

Designs performed using the BS 8110 Beam, Column, or Slabs briefs will be as per BS 8110 with inclusion of these amendments. Note also that the default steel grade is now 500 N/mm² instead of 460 N/mm².

AD.2006-1004.5 Features Affecting the Piping Mode

STAAD.Pro can utilise the pipe layout and reactions created in the applications ADLPipe or AutoPipe. The pipe model can be imported in the Piping Mode. The following describes the method of using the Piping Mode and features recently added to this mode.
AD.2006-1004.5.1 Update of Importing Piping Models

The import stage of the Piping Mode has been enhanced by allowing the user to better relate the co-ordinate systems used by the two models, the structural model and the piping model. There are two issues that are addressed:

a. Locating the origin

b. Allowance for Y or Z axis as vertical in the piping model

Enter the Piping Mode by either selecting the menu option **Mode > Piping** or from the Mode Bar thus:

This displays the principal page and two sub pages available in this mode thus:

To select a pipe model to import, click on the **Open File...** button in the Information dialog box and navigate to the folder that has the required ADLPIPE or AutoPipe (*.ADI) model. Select the required file and click on the **Open** button. The following dialog is then displayed:

This allows the direction that was used as vertical in the pipe model system to be identified by clicking the option **Consider Z as the gravity axis for the Pipe Model** if required. Additionally, the X, Y, and Z values allows the origin of the pipe model to be placed at a different location than the origin of the STAAD.Pro structure.

Refer to the **Piping Mode section** (on page 878) in the User Interface help

AD.2006-1004.5.2 Pipes Support by Member

Supports defined in the piping model may not have corresponding nodes defined in the STAAD.Pro model. Therefore the pipe support function has been enhanced to allow members to be selected so that the support load will be transferred as a member load rather than node load.

Enter the Piping Supports page. Select the menu item 'Support>Support Node'. Using the Pipe Support cursor, select the pipe support node then click on the supporting member.
AD.2006-1004.6 Additional STAAD.Pro 2006 Build 1004 Features

The following pages outline the additional features that have been added to STAAD.Pro in this version.

AD.2006-1004.6.1 Modifications of the SELECT XM Security System

With the SELECT security system, STAAD.Pro can be configured to use either a **STAAD.Pro** license or a **Universal Structural License** (USL), which can also be used to access other structural products in the Bentley portfolio. However, to perform a design, the USL needs to be configured along to specify which design code packs are to be accessed during the operation of STAAD.Pro. Additionally, once the configuration of the STAAD.Pro SELECT license has been established, it is now possible to review the current settings and update it as required.

To review and/or update the current STAAD SELECT security configuration, click on the Start menu item: **Start > Programs > STAAD.Pro 2006 > Select License Tools > STAAD SELECT Configuration**
1. Select STAAD.Pro from the Product drop list.
2. Select whether a Standard or USL license should be obtained to run STAAD.Pro.
3. Select which code packs should also be obtained when STAAD.Pro is run.

Notes:

1. This configures what licenses are to be obtained. This does not confirm that those features are available. To confirm what features are available, click Settings. This launches the License Management Tool which displays the list of products and licences available on the currently defined SELECT XM server and those that are currently 'Checked Out' onto the local machine.
2. If a feature or design code requires a specific license during a session of STAAD.Pro, then it is important to check that the license for that feature is available using the License Management Tool and setting the STAAD SELECT XM License Configuration to obtain that license as STAAD.Pro starts.

For a complete description of the SELECT security system, please refer to the Quickstart & Troubleshooting Guide (STAAD.Pro_InstallGuide_en.pdf) document distributed with the installation setup.

AD.2006-1004.6.2 AISC N690

In addition to the standard ASD design method, STAAD.Pro can perform a design to incorporate the design requirements of Supplement No. 1 to the Specification of the Design, Fabrication and Erection of Steel Safety-Related Structures for Nuclear Facilities ANSI/AISC N690 – 1994s1. This additional code has been added to the Steel Design dialog.

To perform a design of steel sections to the AISC N690, a design parameter block containing the command CODE AISC N690 followed by the command to perform a CODE CHECK or MEMBER SELECTION must be added to the STAAD.Pro data file.

The GUI has been updated by adding AISC N690 in the Steel Design dialog:
When this design code option is selected, clicking on the Define Parameters... button at the base of the dialog box will in turn open the Design Parameters dialog that will allow parameters to be added or added and assigned to specific steel members.

For a full explanation of the features of the design engine see D1.K. American Codes - Steel Design per ANSI/AISC N690 Design Codes (on page 1575).

**Note:** Design to this module is only possible if your security allows design to US Specialised design codes.

**STAAD.Pro 2006 Build 1002 Release Report**

The Software Release Report for STAAD.Pro 2006 Build 1002 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 build 1001. This document should be read in conjunction with all other STAAD.Pro help, including the Revision History document.

**AD.2006-1002.1 Features Affecting the Pre-Processor (Modeling Mode)**

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.

**AD.2006-1002.1.1 Automatic calculation of the Response Spectrum as per IBC**

STAAD.Pro previously allowed response spectra to be defined by the user, or calculated as per the Indian IS 1893 or Eurocode EC8.

The graphical user interface (GUI) has been enhanced so that it can now automatically generate a response spectrum as per the guidelines of the IBC/ASCE code.
(*) This new feature saves users the time and effort involved in establishing the 1 second and short period accelerations indicated in the IBC/ASCE maps and then converting them into spectral data based on the equations provided in the code.

The user enters the latitude and longitude (or select a US zip code), STAAD.Pro then establishes the site coefficients, S1 and SS. The user then selects the appropriate site class (A-F) and with this information, STAAD.Pro calculates the MCE spectral acceleration and design spectral acceleration as per sections 11.4.3 and 11.4.4 of the ASCE7-05 code. Finally the design response spectrum will be generated based on section 11.4.5 of the ASCE7-05 code.

Note: There is no change to the input command structure.

Description

Go to the General | Load page and in the Load dialog box, open the Load Case Details and select the appropriate primary load case with the masses (defined as load items) defined or create a new primary load case and add the masses, if required. Click on the Add button which will bring up the Add New: Load Items dialog box. Select the Response Spectrum option from the list on the left.

With the code option set to Custom, a Generate IBC Spectrum button will now be available as shown thus

Click on the Generate IBC Spectrum button to open the Spectrum Parameters dialog box thus
Refer to **Spectrum Parameters dialog** (on page 3027) for details on the input fields in this dialog.

To complete the generation of the IBC response spectrum pairs, click on the Close button. The completion of the Response Spectrum command continues as for all other Custom defined Response Spectrum commands. Note that the details of the spectrum Period, Acceleration pairs are displayed in the Response Spectrum window, but cannot be modified at this time.

For more details on the Response Spectrum loading command, refer to Section 5.32.10.1 of the Technical Reference Manual (on page 2688).

**Related Links**

- *M. To add an IBC 2000 response spectrum* (on page 844)

**AD.2006-1002.1.2 Enhancement to Physical Member Query Window**

With the enhancement of the physical Member in the Pre-Processing Mode, the Query dialog box obtained when a physical member is double clicked.

Select a Physical Member by using the Physical Member cursor, \[\text{Physical Member cursor} \] and double-click on that to view the **Physical Member Query** dialog box.

**Pre-Processing Information**

Prior to any analysis being performed, the physical member information available is displayed in the following three sheets **Geometry**, **Property**, and **Loading**.

All those common physical member attributes, assigned to that particular physical member (i.e., to all the analytical members comprising that physical member) are displayed in these information fields, as shown below:
Post Processing Information

If an analysis has been performed and there are results available, the following post processing results are made available in two additional sheets Shear Bending, to view the Shear Force diagram, Bending Moment Diagram, and Deflection to display deflected shape information.

Note: The design of Physical Members is only performed in the Steel Design Mode and note the batch method in the current version of STAAD.Pro.

Related Links
• Physical Member dialog (on page 3190)
AD.2006-1002.1.3 AASHTO (LRFD) Design Code

The STAAD.Pro analysis and design engine has been enhanced with the addition of a new design code, AASHTO (LRFD). The Graphical User Interface (GUI) has been updated to allow the design parameters to be entered and applied as required.

Go to the Design | Steel page and in the Steel Design dialog box, select the option AASHTO (LRFD) from the Current Code drop-list. Click on the button Select Parameters to display the parameters available to be used with this design code. With all parameters displayed in the right Selected Parameters’ panel, close the dialog box. Click on the Define Parameters to set the values for the parameters which can be added or assigned to selected members.

Click on the Commands button to add or assign the commands to the file to instruct a check or selection of sections based on the parameter settings.

The AASHTO design parameters listed in previous versions of STAAD.Pro as AASHTO in the Current Code drop-list of the Steel Design dialog box, have been renamed as ‘AASHTO (ASD)’

Note that access to design codes is controlled by your licence agreement and security settings. Not all users have access to all design codes. If this code is unavailable, see your STAAD.Pro supplier to obtain the updates needed to activate this or any other design code.

AD.2006-1002.1.4 Node Precision

A number of models require a higher than usual precision on the node co-ordinates that are used by the analysis engine.

With STAAD.Pro running, but without having any structural model loaded, select File > Configure.

In the resulting dialog, click on the Input File Format sheet, there is a new section Significant Digits which allows the required value to be set.

Related Links
**What's New?**

STAAD.Pro 2006 Release Reports

- *GS. Application Configuration dialog* (on page 55)

**AD.2006-1002.1.5 User Steel Table - Double Angles**

To design a beam that has been defined as a Double Angle from a User table, then an 11th parameter must be provided in the section parameters which defines the RVV value. The GUI has been enhanced to support this additional parameter.

**Description**

Select **Tools > Create User Table**.

Start a new table which is of type **DOUBLE ANGLE** and click on the **Add New Property** button which will display the following dialog which allows the value of Rvv to be included in the section property definition.

![Double Angle Table](attachment:image)

**Related Links**

- *TR.19.4 Double Angle* (on page 2452)

**AD.2006-1002.2 Features Affecting the Analysis and Design Engine**

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

**AD.2006-1002.2.1 AASHTO Design Code**

The STAAD.Pro analysis and design engine has been enhanced with the addition of a new design code, AASHTO (LRFD).

**AD.2006-1002.2.2 Updated Parameters on Indian IS 1893 Static Seismic Loading Command**

The static equivalent method for creating loads as per the Indian seismic code, IS 1893 has been enhanced with the option for checking soft stories.

**Related Links**

AD.2006-1002.2.3 Updated Parameters on Indian IS 1893 Response Spectrum Command

The method for creating a response spectrum to the Indian seismic code, IS 1893 has been enhanced with the option for checking soft stories and documentation on the torsion option added.

Related Links
- TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2721)

AD.2006-1002.3 Features Affecting the Steel Design Mode

Several new features have been added and existing features have been modified in the Steel Designer section of the program, also known as the Steel Designer Mode. These are explained in the following pages.

AD.2006-1002.3.1 Access to Physical Members Created in Pre-Processing Mode

Physical members can be created in the Pre-Processor mode and their definition stored in the STAAD.Pro input file (*.STD). Physical members that have been created in the Pre-Processing Mode are now available in the Steel Designer Post Processing Mode. Previous versions of STAAD.Pro would require these members being re-defined in the Steel Designer Mode.

Create Physical Members as defined in the manual AD.2006.1.1 Physical Member Interface (on page 313). Perform an analysis and enter the Steel Designer Mode. Go to the Member Design | Member Setup Page, and note that the members that were created in the Pre-Processing Mode are ready to be utilised in the Steel Designer Mode.

AD.2006-1002.4 Features Affecting the Concrete Design Mode

Several new features have been added and existing features have been modified in the RC Designer section of the program, also known as the Concrete Design Mode. These are explained in the following pages.

AD.2006-1002.4.1 DXF Output of Beam and Column Designs

It is now possible to write the completed design drawing to a raw DXF file, as well as to MultiREBAR format, so that it may be finished off by a detailer. The MultiREBAR format drawing requires a suitable installation of the MultiSuite application to visualize the drawing. The raw DXF format can be viewed by any application that can view DXF files.

Click on the Drawing or Schedule page (according to design code), a dialog opens as shown below, except for the addition of a drop list to choose the output format.
The MultiREBAR export remains as before, the DXF format is new. For DXF output, the concrete outlines, covers and center lines are all output as simple lines to appropriate layers. The bars are sent as polylines, again to separate layers. The schedule is not exported, but may be printed from the normal report setup. In all other respects the dialog works in the same way as described in the RC Design help (on page 1188).

Related Links
•  D. Export to Drawing dialog (on page 1188)

AD.2006-1002.5 Features Affecting the Advanced Slab Design Mode
The Mode previously known as the ADAPT Slab design has been redefined as the Advanced Slab Design and has been enhanced to support integration with RAM Concept.

Note: Beginning with STAAD.Pro CONNECT Edition, direct export to ADAPT-Builder® has been deprecated.

AD.2006-1002.5.1 Advance Slab Design Mode
Refer to the Advanced Slab Design Mode section (on page 1353) of the User Interface help for details.

AD.2006-1002.6 Features Affecting the Piping Mode
STAAD.Pro can utilise the pipe layout and reactions created in the applications ADLPipe or AutoPipe. The pipe model can be imported in the Piping Mode. The following describes the method of using the Piping Mode and features recently added to this mode.

AD.2006-1002.6.1 Piping Mode
Refer to the Piping Mode section (on page 878) in the User Interface help.
AD.2006-1002.6.2 3D Rendered View of Piping Model

In order to perform quick, visual clash detections, the 3D properties of the piping model have been added to the 3D section view and Rendered 3D View.

To view the solid 3D shapes in the graphical window, select the menu option, View > Structure Diagrams... This opens the Diagrams dialog in the Structures sheet. In the 3D Sections group, select the option ‘Full Sections’ to display a solid 3D shape of the structure model and piping arrangement, thus:

To view a rendered 3D view of the structural model with the piping arrangement, select the menu option, View > 3D Rendering... This creates a new window and displays a 3D perspective view of the model and piping thus:

Note: The piping model is only displayed in the Piping Mode.

AD.2006-1002.7 Additional STAAD.Pro 2006 Build 1002 Features

The following pages outline the additional features that have been added to STAAD.Pro in this version.
AD.2006-1002.7.1 Modifications in REI Security System

For details on the current security system for STAAD.Pro, please refer to the STAAD.Pro Installation Guide (STAAD.Pro_InstallGuide_en.pdf).

AD.2006-1002.7.2 Online Help

In order to assist the user operate STAAD.Pro, a number of context sensitive links have been added to the GUI such that clicking on a Help button or pressing the <F1> key will go directly to an appropriate help topic within the online documentation.

STAAD.Pro 2006 Build 1001 Release Report

The Software Release Report for STAAD.Pro 2006 Build 1001 contains detailed information on additions and changes that have been implemented since the final build of STAAD.Pro 2005. This document should be read in conjunction with all other STAAD.Pro help.

AD.2006.1 Features Affecting the Pre-Processor

Several new features have been added and existing features have been modified in the pre-processor section of the program. They are explained in the following pages.

AD.2006.1.1 Physical Member Interface

STAAD.Pro will allow grouping analytical predefined members into physical members using a special member group PMEMBER. PMEMBER defines a group of analytical collinear members, with same cross section and material property.

To model using PMEMBER, one needs to model regular analytical members and the start grouping those together.

While creating a PMEMBER, the following are the pre-requisites,

- Existence of the analytical members in the member-list.
- Selected members should be interconnected.
- The selected individual members are collinear.
- Local axis of the individual members comprising the physical member should be identical (i.e., x, y and z are respectively parallel and in same sense).
- A member in one Physical Member Group should not occur in any other Physical Member Group.

Related Links

- TR.16.2 Physical Members (on page 2443)

General Format

PMEMBER can be created either in modeling mode or Steel-Designer mode. Modeling mode and Steel-Designer mode physical members will be labeled as M and D, respectively. Modeling mode PMEMBER will allow variable cross-sections. Steel-Designer mode will allow importing PMEMBER-s created in modeling mode.

General Format
Following STAAD commands related to PMEMBER are implemented.

```
DEFINE PMEMBER
PMEMBER PROPERTY
PMEMBER CONSTANT
PMEMBER LOAD
PRINT PMEMBER FORCE
```

To define a Physical Member, the following command is used after the MEMBER INCIDENCE Command:

```
DEFINE PMEMBER
member_list PMEMBER pmember-no
```

Example

```
JOINT COORDINATE
1 0 0 0 6 10.0 0 0
MEMBER INCIDENCE
1 1 2 5
DEFINE PMEMBER
1 TO 5 PMEMB 1
```

To define the member property of a Physical Member, the following command is used:

```
PMEMBER PROPERTY
pmember-list PRIS ...
```

The Physical Member supports all types of member properties available in STAAD.

If multiple definitions of member properties for a particular analytical member is encountered (e.g. analytical member properties is defined twice, once via PMEMBER PROP command and again via MEMBER PROP command, then MEMBER PROP command will override PMEMBER PROP definition.

To define the Material constants of a Physical Member, the following command is used:

```
PMEMBER CONSTANT
E CONCRETE pmember-list
DEN CONCRETE pmember-list
...
```

Any member, which is a part of any PMEMBER is not allowed to be assigned constants explicitly.

**Note:** Loads are applied directly to PMembers using the PMEMBER LOAD command. See “PMember Load Specification” for details.

After the analysis, the Post Analysis results of a PMEMBER can be seen by using the following command:

```
PRINT PMEMBER FORCE
```

This command will produce member forces for all the analytical members in the group.

**Graphical User Interface for Physical Member**

The following steps are to be followed for creation of a Physical Member

Post-Processing Features Related to Physical Members

In the post processing results, to view the Shear Force diagram, Bending Moment Diagram and Deflected shape, use the member query dialog box.
Select a Physical Member by using the Physical Member cursor, and double-click on that to view the member query dialog box.

Design of Physical Members

In the current version of STAAD.Pro, the design of a Physical Member is not handled.

To Create a Physical Member

Make a frame model in STAAD.Pro, comprising of two or more consecutive beams.

1. Select two or more colinear, interconnected beams in the model.
   Refer the pre-requisites of the analytical members for details.
2. Either:
   - Select the **Form Member** tool in the **Physical Member** toolbar
   - Right-click and select **Form Member** from the pop-up context menu.

The details of the formed Physical Member can be seen in the **General | Physical Member** page at the right pane of the window.

To assign properties to a physical member

You must first create one or more physical member properties and then these can be assigned to physical members.

1. Select the **General | Property** page.
2. Select the **Toggle Physical Member mode** tool in the **Physical Member** toolbar.
3. Create a physical property:
   a. In the **Properties - Whole Structure** dialog, click **Define**.
      The **Property** dialog opens.
   b. Select a property class and define the required property parameters.
   c. Click **Add**.

A new property is added to the list in the **Properties - Whole Structure** dialog. This property is marked as “Physical”.
4. Assign this property to the physical member.

**Note:** For a Physical Member whose physical member property is already assigned, the individual analytical members in that Physical member will adopt the same member property that of the physical member. However, in case, where the analytical member property is assigned to any member in that physical member, then this analytical member property will supersede physical member property.

To assign specifications to a physical member

Only the Release, Offset, and Truss member specifications are available for physical members.

1. Select the **General | Spec** page.
2. Select the **Toggle Physical Member mode** tool in the **Physical Member** toolbar.
3. Create the member specification:
   a. Click **Beam** in the **Specifications - Whole Structure** dialog.
      The **Member Specification** dialog opens.
b. Select the dialog tab corresponding to the specification type you want to add.
c. For Release or Offset specifications, specify the parameters to define the specification.
d. Click Add.

A new specification is added to the list in the Specifications - Whole Structure dialog. Release and Offset specifications are marked as "Physical" whereas Truss specifications are labeled PMEMBER TRUSS.

4. Assign this specification to the physical member.

**Note:** For a Physical Member whose physical member specification is already assigned, the individual analytical members in that Physical member will adopt the same member specification that of the physical member.

However, in case, where the analytical member specification is assigned to any member in that physical member, then this analytical member specification will supersede physical member specification.

To assign loads to a physical member

1. Select the General | Load & Definition page.
2. Select a load case in the Load & Definition dialog and click Add.

Physical member loads can be assigned to a new or existing load case.

The Add New : Load Items dialog opens.

3. Select the Physical Member Loads tab.
4. Specify the load item parameters for a load type and click Add.

The load item is added to the load case and is marked as "Physical".

5. Assign the physical load items to physical members.
**Note:** For a Physical Member whose physical member Load is already assigned, the individual analytical members in that Physical member will adopt the member loads internally depending on the physical member load. However, in case, where the analytical member Load is assigned to any member in that physical member, then this analytical member Load will superimpose on the physical member Load.

### AD.2006.1.2 Load Envelopes

Load page will allow creation of Envelopes that can be saved as part of the STAAD input file. The user will be able to define multiple load envelopes each consisting of groups of predefined load cases. These envelopes can latter be used for post-processing. For example post analysis results may be viewed for a selected load envelope. As far as the STAAD engine is concerned, ENVELOP command will translate in to a LOAD LIST command.

The envelope can be tagged with optional key words to specify qualitative nature of the load or load combination cases included in the envelop definition. Based on the nature of the load cases in the envelope, the users can define appropriate design parameters for each envelope. For example, for design under wind load condition, most of the design codes allow increase of allowable stresses. Design routine can increase the allowable stress used in interaction equation, when it does the design for the envelope. Another application of this feature can be to specify separate load groups for serviceability check, working stress and limit state checks.

### Description

The option for defining **Load Envelopes** appears within the **Load** dialog box in the **General | Load** page as shown in the next figure.

![Load Envelopes Dialog Box](image)

To define an envelope, select the Load Envelopes and click on the Add button and the Add New : Load Envelopes dialog box will come up as shown in the next figure.
The Envelope will be identified by the number that appears in the Envelope edit box. One can select an appropriate type for the load envelope like stress, serviceability etc. depending on the nature of the loads selected for the envelope. All the predefined primary load cases and combination cases will appear inside the Available window. One can now select one or more of these cases, bring them to the Selected window on the right hand side and click on the Add button to create the envelope. Once an envelope is created, it will be displayed in the Load dialog box as shown in the next figure.

In the post processing mode, the Results Setup dialog box will include option for selecting Defined Envelopes for the purpose of displaying results as shown in the next figure. Once an envelope is selected the corresponding loads can be seen within the Selected window.
General Format

The load envelope commands are written in the STAAD input file as shown below.

```
DEFINE ENVELOP
  load-case-list ENVELOP # TYPE { NONE | STRESS | SERVICEABILITY | COLUMN | CONNECTION | STRENGTH }
END DEFINE ENVELOP
```

**Example**

```
DEFINE ENVELOP
  1 2 ENVELOP 1 TYPE SERVICEABILITY
  3 5 ENVELOP 2 TYPE STRESS
END DEFINE ENVELOP
```

The first line within DEFINE ENVELOPE command means that load cases numbered 1 and 2 make up the serviceability type load envelope 1. Similarly load cases 3 and 5 define the stress type load envelope 2.

To print out the support reactions corresponding to load envelope 1, that includes load cases 1 and 2, the following commands will be defined in the input file

```
LOAD LIST ENV 1
PRINT SUPPORT REACTIONS
```

If ENV keyword is encountered, the list will be interpreted as list of envelopes rather than a list of load cases.

**Note:** Please refer to the example file Load_envelopes.std, which is available under the different country folders.

**Related Links**

- *TR.40 Load Envelope* (on page 2837)
AD.2006.1.3 Persistency of Parametric Mesh Model in STAAD Input File

In the past, once the parametric mesh model was merged with the base model, no information about the parametric mesh was retained by STAAD. So, if any modification was required at a later stage, the parametric mesh had to be created afresh. Parametric model feature has now been enhanced and multiple parametric mesh models can now be saved as part of the STAAD model. This would allow the users the flexibility to come back to the saved mesh models at any time and make modifications to it like adding an opening or adding a density line.

The parametric mesh model data is now saved as part of the STAAD input file. Special tag based commands has been introduced to support saving of parametric mesh models as part of the STAAD input file as shown below.

```
2072 1114 1113 1160; 2073 1045 1160 1113;
ELEMENT PROPERTY
  810 TO 1779 1821 TO 2073 THICKNESS 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
PARAMETRIC MODEL SLAB
MESH PARAM 0 3
MESH ORG 3 5 8
BOUNDARY 10
  11 1 93 1 94 1 95 1 83 1 71 1 70 1 69 1 41 1 26 1
OPENING CIRC 72 360 96 43.2666 12
OPENING POLY 5
  216 360 67.2 1 270 360 33.6 2 324 360 67.2 2 270 360 100.8 2 216 360 100.8 2
DENSITY POINTS 2
  180 360 168 1 360 360 168 1
DENSITY LINE 0 360 168 100 180 360 168 200
DENSITY LINE 180 360 168 1 360 360 168 1
DENSITY LINE 360 360 0 1 180 360 168 1
DENSITY LINE 180 360 168 1 180 360 336 1
DENSITY LINE 360 360 0 1 360 360 168 1
DENSITY LINE 360 360 168 1 360 360 336 1
DENSITY LINE 54 360 302.4 1 162 360 201.6 1
DENSITY LINE 216 360 201.6 1 324 360 235.2 1
GENERATED PLATES ALL
END
<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
PARAMETRIC MODEL FIRST_FLOOR_SLAB
MESH PARAM 60 3
MESH ORG 2 3 5
BOUNDARY 6
  36 1 65 1 66 1 53 1 52 1 51 1
GENERATED PLATES ALL
END
!> END GENERATED DATA BLOCK
!> END GENERATED DATA BLOCK
DEFINE MATERIAL START
ISOTROPIC STEEL
```

Go to the Geometry | Parametric Models page and the saved parametric mesh models will appear within the Parametric Models dialog box as shown in the next figure.
There are two parametric mesh models named `second_floor` and `roof` inside the **Parametric Models** dialog box as shown in the previous figure.

**Note:** Please refer to the example file `parametric_models.std`, which is available under various country folders.

**Related Links**
- **Parametric Models dialog** (on page 2910)

**AD.2006.1.4 Persistency of Parameters User to Generate ASCE Wind Load in STAAD Input File**

In the past, once the wind load as per the ASCE-7 code was generated automatically using the wind load generator, no information about the parameters was retained by STAAD. So, there was no way to check the parameters, based on which the generation was done. Moreover, if any modification was required at a later stage, all the parameters had to be defined afresh. Automatic wind load generation feature has now been enhanced and the parameters can now be saved as part of the STAAD.Pro model.

Special tag based commands has now been introduced to support saving of ASCE parameters as part of the STAAD input file as shown below.

```
DEFINE WIND LOAD
  TYPE 1
  INT 0.017517 0.017517 0.0176388 0.017754 0.0178632 -0.0179672 0.0180665 0.0181615
  0.0182526 0.0183402 0.018506 -0.0184246 0.0185846 0.0186607 0.0187345 -
  26.5385 27.6923 28.8462 30

<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2002:PARAMS 85.000 MPH 0 2 2 0 0.000 FT 0.000 FT -0.000 FT 1 1 30.000 FT
```
When a user goes back to the Load page and goes to Definitions | Wind Definitions page and double clicks on the Intensity as shown in the next figure,

the intensity versus height data shows up as shown below.

One can now click on the Calculate as per ASCE-7 button and the parameters that were initially defined are going to appear in the resulting dialog boxes.

Note: Please refer to the example file ASCE_WIND_load_generation.std, which is available under various country folders.

Related Links
- TR.31.3 Definition of Wind Load (on page 2623)
- ASCE 7 Wind Load dialog box (on page 3039)
- Wind Load tab (on page 3018)
- M. To add an ASCE 7 wind load definition (on page 836)

AD.2006.1.5 Enhancement of Z UP System

Earlier, when geometry created using a CAD software that used Z axis as the vertical axis, had to be imported into STAAD.Pro, the users had to reorient the model to make the Y axis the vertical axis before getting the model into STAAD.Pro. This was necessary because a number of STAAD.Pro load generation commands did not work.
when the Z axis was defined as the vertical axis. In STAAD.Pro 2006 these limitations has been addressed. All the load generations are going to work with the Z axis up coordinate system.

The default coordinate system for a model can be set by going to File > Configure. The Configure option is only available when no STAAD model is open. If a STAAD model is open, one has to click on File > Close to close the file first.

The Configure Program dialog comes up as shown in the next figure.

The Z up option can then be selected and applied to the model. This would set the Z axis to be the vertical axis instead of the default Y axis. Subsequently all modeling, analysis and postprocessing items would be based on this coordinate system.

Related Links

• GS. To use Z as the vertical axis (on page 46)

AD.2006.1.6 Specification of Member Orientation Using Reference Vector

This feature will allow users to orient members by specifying a direction vector. Beta angles will be calculated by the software by itself based on the direction vector specification. The new reference vector will be defined with respect to the local coordinates of the member. This is going to make the task of orienting members much easier.

To access the feature go to Commands > Geometric Constants > Member Reference Point.

The Reference Point dialog box comes up where the vector for the reference axis can be specified as shown in the following figure. The local Y axis for the member is going to oriented along the vector. This X, Y, Z values are going to be based on the local axis system of the member.
The feature is explained in the next section with an example. Let us consider the beam shown in the next figure. The requirement is to rotate the cross section about the axis of the member in the direction as shown, such that the angle of rotation is governed by a slope of 1 horizontal to 2 vertical.

1. Select Commands > Geometric Constants > Member Reference Point. The Reference Point dialog opens.
2. Select the Vector option and specify the reference vector so that it is oriented as per the required slope.
3. Click OK. The member is rotated to the reference vector.
The member gets oriented correctly without the user having to take the trouble of calculating the beta angle. In the STAAD input file, the following command lines get written.

```
CONSTANTS
  REFSVCTOR 0 2 1 ALL
```

Beta angle for the member can be easily figured out by simply double clicking on the member that brings up the **Beam** dialog box as shown next. The correct beta angle of 26.5651 is automatically calculated by STAAD.Pro based on the reference vector specification.

![Beam dialog box](image)

**Related Links**

- **TR.26.2 Specifying Constants for Members and Elements** (on page 2503)
- **M. To align a member to a reference point** (on page 796)

**AD.2006.1.7 Single File Archive to Save All STAAD Input & Output Files**

Based on user configurations, file open and save dialogs will allow user to select .stz (std archive) files. The archive will be expanded to TEMP folder or user configurable working folder. All input/output will be created at that location.
While exiting the program/session, all of those files will be archived again under same name and copied to original source folder.

Archive related commands can be found on the File menu.

**AD.2006.1.8 Auto Calculation of Effective Length Factors of Members as per AISC Code**

STAAD.Pro has a new feature to auto-calculate the Effective Length Factors (KY, KZ etc), for the members to be designed as per AISC-ASD Code.

To make use of this feature, the user needs to go to the Design | Steel page. Select the Country Code as AISC ASD. Select the member for which the Effective Length Factors are to be calculated. Click on the Define Parameters button and select the parameter KY or KZ.

The following dialog box will appear.
The only difference from the previous version of STAAD.Pro is the addition of the Calculate button as marked in the figure.

On clicking the Calculate button, the program will ask whether the braced or un-braced effective length of the selected member is required.

After getting confirmation from the user, the program will calculate and display the value of the Effective Length Factor in the Edit box.

Now, click on the Assign button to associate this value of the respective Effective Length Factor with the selected member.

Note: The calculation of the effective length factors are done as per a paper titled Compression Members presented by George Tsiatas.

AD.2006.2 Features affecting the Post-Processor (Results Mode)

Several new features have been added and existing features have been modified in the post-processor section of the program. They are explained in the following pages.

AD.2006.2.1 Generation of Transfer Force Report for Connection Design

STAAD.Pro can now calculate the “Transfer force” or “pass through force” that can be used for connection design. This feature is based on a paper on the subject by Dr. William A. Thornton. Refer to the next figure which shows beams and bracing members connected to either side of the column.
Transfer force is simply the maximum net horizontal force that gets transferred from the one side of the column to the other through the connection. So STAAD.Pro checks the forces in the members framing into each side of the column and finds out the resultant horizontal force for either side. Typically the resultant forces on the two sides would not be equal as some amount of force will be taken up by the column in shear. The greater of the two resultants is reported by STAAD.Pro as the transfer force. The option to determine transfer force automatically, will save engineers considerable time and effort as in most cases, they have to report the transfer forces in the design drawings. The same concept can be applied to floor bracing in horizontal plane.

**Description**

Go to the Post processing mode by selecting **Mode | Postprocessing** or by clicking on the **Postprocessing** tab above the graphics window. Select all the members that frame into any connection as shown in the previous figure. Then select **Report > Column Transfer**.

A dialog box titled **Transfer Force for Selected Members** comes up as shown in the next figure. The **Loads** box displays the load cases that has been considered to calculate the transfer force. By default all load cases are considered but one can exclude a few of them by simply clicking on the load case within the **Loads** box. The boxes **Left Beams** and **Right Beams** list all the members on the respective sides of the column along with their incidences. The boxes **Left TF** and **Right TF** shows the resultant horizontal force from either side. **Max TF** reports the transfer force.
Click on the Insert to Table button and a table will be generated containing all the transfer force information as shown in the next figure.

The transfer force information can also be included as part of a report by selecting File > Report Setup or by selecting the Report Setup tool, which will open the Report Setup dialog box as shown in the next figure.
By default, the **Transfer Force Report** appears in the selected list of items within the dialog box. A report with the transfer force data can then be generated as shown next.

![Transfer Force Report](image)

**Note:** Please refer to the example file `TransferForces.std`, which is available under various country folders.

**Related Links**
- *Transfer Forces for Selected Members dialog* (on page 3150)
- *P. To generate transfer forces report* (on page 2219)

**AD.2006.2.2 Customizable Color to Display Unity / Check Utilization Ratio**

View steel design results.

Default color for display of passed members was black. It has now been changed to green. The colors representing the various ranges for utilization ratios can be set by the user.
AD.2006.2.3 Enhancement to Steel Designer BS 5950 Calculation Sheets

The design calculation sheets of a steel section that has been designed with a BS 5950-1:2000 design brief has been modified with the following enhancements.

1. The terminology for elastic and plastic sections updated to use same terminology as in the design code.
2. A summary has been added at the end of the calculation sheet.
3. The section classification summary has been improved.
4. The reported dimensions of the section in the 'Compression Flange Buckling Check' have been improved.
5. The status for various checks has been re-titled.
6. Details of check to 4.8.3.3.1 added when required.
7. There are improved details of the clauses 4.8.2., 4.8.3. and Annex I1 calculations.
8. The 4.3.6. Lateral Torsional Buckling Check has been enhanced.
9. Additional detailed calculations for axially loaded members have been added to the document.
10. A number of spelling issues have been addressed.

AD.2006.3 Features affecting Analysis and Design

Several new features have been added and existing features have been modified in the analysis and design section of the program. They are explained in the following pages.

AD.2006.3.1 Pushover Analysis

This is a set of procedures to implement a Pushover Analysis as defined in the document FEMA 356:2000.

STAAD Pushover analysis in STAAD is a static, non-linear procedure in accordance with FEMA 356 specification. Basically, in this method, the magnitude of the lateral push load is increased progressively according to a predefined loading pattern until either loading or the deflection reaches the described level.

Please refer to M. Pushover Loads (on page 861) for details on using this feature.
AD.2006.3.2 Steady State Analysis

Please refer to Section 5.37.6 of the STAAD Technical Reference manual (on page 2807) for details.

AD.2006.4 Features affecting the RC Designer Mode

Several new features have been added and existing features have been modified in the RC Designer mode of the program. They are explained in the following pages.

Refer to the RC Designer Manual for full details. Note that references made here are for sections in the RC Designer Manual.

AD.2006.4.1 Slab Design to BS 8110

This a complete new mode that allows slabs to be defined from a collection of finite elements created from user defined meshes or surface objects. This release has added the design of slabs to BS8110.

See also

- General Overview, 1.3.3 Slab Design Briefs
- Getting Started, 2.6 Slabs Page, 2.8.4.5 SB8110 Slab
- Engineering Information, 3.4.4.3. BS8110 Slab Design Principles

Related Links

- D. BS8110 Slab Design Brief dialog (on page 1260)
- D. Concrete Slab pages (on page 1212)

AD.2006.4.2 BAEL Beam Design Enhancement

The BAEL 91 design code has been updated so that the limitation of reinforcement defined in clause A.8.1.21 is applied to the entire cross section, not just the compression zone of the cross section. This therefore allows for a greater amount of reinforcement in any given section. This new interpretation of the code is more in line with the code requirements in the EC2 and DIN codes.

See also

Engineering Information, 3.4.3.1 BAEL Beam Design Principles

AD.2006.4.3 French GUI

Recent changes to the program's language facilities have deprecated these changes.
AD.2006.4.4 DIN 1045-1 Beam and Column Design

A new DIN 1045-1 set of design codes have been added for the design of beams and columns.

See also

- Getting Started, 2.8.6.1. DIN1045 Beam, 2.8.6.2. DIN1045 Column
- Engineering Information, 3.4.6. DIN1045-1

Related Links

- D. DIN 1045 Beam Design Brief dialog (on page 1274)
- D. DIN 1045 Column Design Brief dialog (on page 1280)

AD.2006.4.5 Use of Generated Load Cases

The RC Designer has been enhanced to make use of the results of loadcases created by STAAD.Pro’s LOAD GENERATION command. These loadcases are now available for use in the Envelope definition, used by beam and slab design briefs and in the loadcases available for column briefs.

AD.2006.4.6 Use of Primary Load Cases in Column Designs

The RC Designer has been enhanced such that all column briefs can include the results of primary loadcases as well as combination loadcases.

AD.2006.4.7 BS8110 Beams Torsion Check Added

The BS 8110 beam design brief has been enhanced with the additional option of identifying the additional reinforcement required for the torsional forces. The additional reinforcement requirement is displayed in the Detail Design option of a user report.

See also

Engineering Information, 3.4.4.1. BS8110 Beam Design Principles

AD.2006.5 Additional STAAD.Pro 2006 Features

The additional enhancements made to STAAD.Pro 2006 are explained in the following pages.

AD.2006.5.1 STAAD.Pro Language Application

Recent changes to the program’s language facilities have deprecated these changes.

AD.2006.5.2 Section Wizard databases

Section Wizard Section Builder module and FreeSketch modules make use of defined standard steel databases. 3 new additional databases have been added and 3 have been updated.

New Additional Databases

- INDIAN.PRF - Indian
- JAPANESE.PRF - Japanese
SAFRICA.PRF - South African

Updated Databases

    BRITISH1.PRF – UK Corus
    DIN.PRF - German
    USA.PRF - ASTM

Note, all other databases are unchanged.

AD.2006.6 Features Added in STAAD.Pro 2005 Previously Undocumented

The following pages explain features that have been introduced into STAAD.Pro but were not fully documented in previous Release Reports.

AD.2006.5.1 DESCON, Advanced Connection Design Mode

Recent changes to the program's language facilities have depreciated these changes.

RAM Connection is now the supported facility for steel connection design in STAAD.Pro.

AD.2006.5.2 ADAPT Slab Design Mode

Recent changes to the program's language facilities have depreciated these changes.

Refer to the User Interface help for use with the Advanced Slab Design mode (on page 1353).

Note: Beginning with STAAD.Pro CONNECT Edition, direct export to ADAPT-Builder® has been deprecated.

AD.2006.6.3 BS5950 Part 5 - Cold Formed Steel Design

Refer to D3.E. British Codes - Design per British Cold Formed Steel Code (on page 1682) for details.

AD.2006.6.4 EC4 Timber Design

Refer to D5.E. European Codes - Timber Design Per EC 5: Part 1-1 (on page 1856) for details.

AD.2006.6.5 Canadian Timber Design

Refer to D4.D. Canadian Codes - Timber Design per CAN/CSA-086-01 (on page 1720) for details.

AD.2006.6.6 South African Steel Design

Refer to D14.B. South African Codes - Steel Design per SANS10162-1:1993 (on page 2181) for details.

AD.2006.6.7 South African Concrete Design

Refer to D14.A. South African Codes - Concrete Design per SABS-0100-1 (on page 2176) for details.
AD.2006.6.8 EC8 Earthquake Loading
Refer to TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994 (on page 2710) for details.

AD.2006.6.9 Additional Kingspan Cold Formed Steel Database
Refer to TR.20 Member Property Specification (on page 2459) for details.

AD.2006.6.10 Imperfection Analysis
Refer to TR.26.6 Member Imperfection Information (on page 2511) for details.

AD.2006.6.11 Tapered Steel Design Added to BS 5950
Refer to D3.B. British Codes - Steel Design per BS5950:2000 (on page 1655) for details.

AD.2006.6.12 User Tools
The users can utilize their own customized tools, in form of VBS Macro, to operate on STAAD.Pro models.

The procedure of associating a user tool in STAAD.Pro is furnished below:

1. Run STAAD.Pro and Open an input file.
2. Select Tools > Configure User Tools.
   The Customize User Defined Tools dialog opens.

3. In the Menu-Items, add a name of the Tool, and in the Command, locate the Tool (VBS Macro) using the browser. Click OK to confirm.
4. Now the Tool name will appear in Tools > User Tools and also by selecting the Tools tool.
5. To Run the Tool (Macro), click on the Tool name in any of the places as described in step 4.

Related Links
• OS. To add the macro to the list of user tools (on page 5075)
STAAD.Pro 2005 Release Report

This Software Release Report for STAAD.Pro Version 2005 contains detailed information on the additions and enhancements made to the program since the final build of STAAD.Pro 2004.

AD.2005.1 Features affecting the Pre-Processor

Several new features have been added and existing features have been modified in the pre-processor section of the program. They are explained in the following pages.

AD.2005.1.1 Generation of Wind Pressure profile per ASCE 7-02

STAAD.Pro is now capable of generating the wind pressure profile for a structure in accordance with the ASCE-7-02 code. The pressure profile is the table of values of wind intensity versus height above ground.

The calculated pressure may then be applied on the structure to compute loads on members using the program’s built-in wind load generation algorithm for the closed as well as open-lattice type structures.

Description

The steps required to generate the wind pressure profile are as follows:

In the General | Load page (from the vertical tabs on the left-hand side), go to the Load dialog box on the right and select Wind Definitions as shown in the next figure.

Click on the Add button to instantiate the Add New Wind Definitions dialog box shown in the next figure. Enter the Type No., which denotes a number by which the wind load type will be identified. Multiple wind types can be created in the same model. Click on the Add button within this dialog box and then click on Close.
The newly created TYPE 1 wind definition will appear underneath Wind Definitions in the Load dialog box as shown below.

Select the TYPE 1 name in the tree control and click on the Add button. The dialog box shown below will prompt for the pressure profile (intensity) for this wind definition.
As we said earlier, the pressure profile is the table of wind intensity versus height above ground. If we know that, that information can be typed into the box above. But, our goal is to calculate that. Hence, we click on the button Calculate as per ASCE-7.

The ASCE-7: Wind Load dialog box shown below will appear.

The options shown in the dialog box are explained below:

**Common Data**

Depending on which version of the code to use, choose either 1995 or 2002.
<table>
<thead>
<tr>
<th><strong>Building classification category</strong></th>
<th>Building classification category as obtained from Table 1-1 in SEI/ASCE 7-02. Category can be I, II, III or IV.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Basic Wind Speed</strong></td>
<td>Basic Wind Speed as described in section 6.5.4 of the SEI/ASCE 7-02 code.</td>
</tr>
<tr>
<td><strong>Exposure Category</strong></td>
<td>Exposure category as described in section 6.5.6.3 of the SEI/ASCE 7-02 code.</td>
</tr>
<tr>
<td><strong>Structure Type</strong></td>
<td>Select the type of structure that best fits the model from the available choices:</td>
</tr>
<tr>
<td></td>
<td>- Building structures</td>
</tr>
<tr>
<td></td>
<td>- Chimneys, Tanks and similar structures</td>
</tr>
<tr>
<td></td>
<td>- Solid Signs</td>
</tr>
<tr>
<td></td>
<td>- Open Signs</td>
</tr>
<tr>
<td></td>
<td>- Latticed Framework</td>
</tr>
<tr>
<td></td>
<td>- Trussed tower</td>
</tr>
<tr>
<td></td>
<td>The associated input pages will change depending on the selection of the type of structure.</td>
</tr>
<tr>
<td><strong>Consider Wind speed-up over hill or escarpment</strong></td>
<td>If there are isolated hills and escarpments that constitute abrupt changes in the general topography, the increase in speed can be considered as per section 6.5.7 in the SEI/ASCE 7-02 code. Select 'Yes' to consider wind speed-up over a hill or an escarpment and 'No' to ignore it.</td>
</tr>
<tr>
<td><strong>Type of Hill or Escarpment</strong></td>
<td>Select the type of hill, ridge or escarpment on which the structure is located, based on figure 6-4 of the SEI/ASCE 7-02 code. The options available are 2-D Escarpment, 2-D Ridge and 3-D Axisymmetric Hill.</td>
</tr>
<tr>
<td><strong>Height of Hill or Escarpment (H)</strong></td>
<td>Specify the height of the hill or escarpment relative to the upwind terrain (H in Figure 6-4 of the SEI/ASCE-7-02 code).</td>
</tr>
<tr>
<td><strong>Distance upwind of crest (L&lt;sub&gt;H&lt;/sub&gt;)</strong></td>
<td>Specify the distance upwind of crest to where the difference in general elevation is half the height of the hill or escarpment (L&lt;sub&gt;H&lt;/sub&gt; in Figure 6-4 of the SEI/ASCE-7-02 code).</td>
</tr>
<tr>
<td><strong>Distance from the crest to the building (x)</strong></td>
<td>Specify the distance from the crest to the building site. A negative value signifies that the distance is in the downwind direction (X in Figure 6-4 of the SEI/ASCE-7-02 code).</td>
</tr>
</tbody>
</table>
Main Building Data

- **Building Height**: The height above ground to the highest point on the roof surface.
- **Building Length along the direction of Wind (L)**: Length of building measured along the direction of wind.
- **Building Length Normal to the direction of Wind (B)**: Length of building measured normal to the direction of wind.
- **Building Natural Frequency**: Specify the natural frequency of the building to calculate Gust Effect Factor.
- **Building Damping Ratio**: Specify the damping ratio to calculate Gust Effect Factor.
- **Enclosure Classification**: Classify the building as open building, partially enclosed or enclosed as per the provisions of section 6.2 of SEI/ASCE-7-02.
- **Kz**: Velocity pressure exposure coefficient that is calculated by STAAD.Pro as per Table 6-3 of the SEI/ASCE-7-02 code.
- **Kzt**: When wind speedup is considered, Kzt is calculated as per Eq 6-4 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and providing the required value in the edit box for Kzt.
- **I**: Importance Factor I is considered as per section 6.2 of the SEI/ASCE-7-02 code. Users can enter their own value by checking the Use box and typing the required value in the edit box for I.
### What's New?

STAAD.Pro 2005 Release Report

**Kd**

Wind directionality factor is calculated as per Table 6-4 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and typing the required value in the edit box for Kd.

The list of parameters explained earlier corresponds to the structure type called “Building Structures”. When structure type is varied, some of those parameters change. Those parameters that are different are explained below for each type of structure.

#### Structure type: Chimney, Tank, and similar Structures

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Height (H)</strong></td>
<td>Height of the structure as defined by the term ‘h’ in Figure 6-19 of the SEI/ASCE 7-02 code.</td>
</tr>
<tr>
<td><strong>Least Horizontal Dimension (W)</strong></td>
<td>Smaller of the plan dimensions. In case the cross section of the structure in plan is circular, the diameter needs to be specified.</td>
</tr>
<tr>
<td><strong>Horizontal Cross-Section Type</strong></td>
<td>This is the cross section of the structure in plan as defined in Figure 6-19 of the SEI/ASCE 7-02 code. The available options include square with wind being normal to face or acting along the diagonal, hexagonal, octagonal and round.</td>
</tr>
<tr>
<td><strong>Depth of protruding elements such as ribs and spoilers (D’)</strong></td>
<td>For round type cross sections, depth of protruding elements need to be defined which is a measure of the surface roughness as indicated in Figure 6-19 of the SEI/ASCE 7-02 code.</td>
</tr>
<tr>
<td><strong>C_f</strong></td>
<td>Force coefficient that is calculated by STAAD.Pro as per Figure 6-19 of the SEI/ASCE 7-02 code. The parameter is used for calculation of design pressure. If desired, users can enter their own value for C_f by checking the Use box and typing the required value in the appropriate edit box.</td>
</tr>
</tbody>
</table>

#### Structure type: Solid Signs

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Height (H)</strong></td>
<td>Height of the structure which is used for calculating the height to width ratio as defined by the term ‘n’ in Figure 6-20 of the SEI/ASCE 7-02 code.</td>
</tr>
<tr>
<td><strong>Horizontal Dimension of Sign (M)</strong></td>
<td>Horizontal dimension of the solid sign</td>
</tr>
<tr>
<td><strong>Vertical Dimension of Sign (N)</strong></td>
<td>Vertical dimension of the solid sign. If the sign is at the ground level, the height (H) and vertical dimension (N) should both be specified the same value.</td>
</tr>
<tr>
<td><strong>C_f</strong></td>
<td>Force coefficient that is calculated by STAAD.Pro as per Figure 6-20 of the SEI/ASCE 7-02 code. The parameter is used for calculation of design pressure. If desired, users can enter their own value for C_f by checking the Use box and typing the required value in the appropriate edit box.</td>
</tr>
</tbody>
</table>

#### Structure type: Open Signs / Lattice Frame Work

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Ratio of Solid Area to Gross Area</strong></td>
<td>Ratio of solid area to gross area as indicated by the term e in Figure 6-21 of the SEI/ASCE 7-02 code.</td>
</tr>
<tr>
<td><strong>Orientation of the members exposed to wind</strong></td>
<td>The type of member surfaces which are exposed to wind. Select flat-sided members or rounded members in Figure 6-21 of the SEI/ASCE 7-02 code.</td>
</tr>
<tr>
<td><strong>Diameter of typical round member</strong></td>
<td>Diameter for round members as defined by the term ‘D’ in Figure 6-21 of the SEI/ASCE 7-02 code.</td>
</tr>
</tbody>
</table>
Structure type: Trussed Tower

**Horizontal Cross Section**  The type of cross section of the tower in plan as defined in Figure 6-22 of the SEI/ASCE 7-02 code. The available options include square and triangle.

### Building Design Pressure

![Diagram of Building Design Pressure](image)

**Building Wall to generate Wind Load on:**

- **Windward** To generate the design wind pressure for the windward side, choose this option. The pressure will be calculated as per Equation 6-23 of the SEI/ASCE-7-02 code. The relevant equation is also displayed in the box just beneath the radio buttons.

- **Leeward** To generate the design wind pressure for the leeward side, choose this option. The pressure will be calculated as per Equation 6-23 of the SEI/ASCE-7-02 code. The relevant equation will be displayed in the box just beneath the radio buttons.

- **Side Wall** To generate the design wind pressure for the side wall, choose this option. As before, the pressure will be calculated as per Equation 6-23 of the SEI/ASCE-7-02 code. The relevant equation will be displayed in the box just beneath the radio buttons.

- **G** Gust effect factor calculated as per section 6.5.8 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and typing the required value in the edit box for G.

- **Cp** External pressure coefficient. The product of external pressure coefficient & gust effect factor is considered as per Figures 6-11 through 6-17 of the SEI/ASCE-7-02 code. Users can modify the value of Cp by checking the Use box and typing the required value in the edit box for Cp.

- **GCpi** Internal pressure Coefficients as per Figure 6-5 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and typing the required value in the edit box for GCpi.
The table on the right displays the intensities at various heights. Click on OK to arrive at the dialog box shown below.

Click on **Add** to add the load definition.

The defined wind loading can then be applied to the structure following the same procedure as in prior versions of the program. For details on the command syntax for generation of wind loads, refer to section 5.32.12: Generation of Loads from the STAAD Technical Reference Manual and example problem 15 in the Examples manual.

**Note:** The option we encountered earlier regarding the windward side, leeward side and side walls tells us that the pressure profile for each of those has to be individually determined under a unique type number. Thus, generating the profile for the 3 sides of the building constitutes 3 separate steps and thus, 3 separate types. Each type can then be applied with one load case or separate load cases and then applied in the relevant direction with the appropriate direction factor. Examples illustrating wind load generation can be found in the examples manual.

**Related Links**
- **TR.32.12.3 Generation of Wind Loads** on page 2779
- **Wind Load tab** on page 3018
- **Add New Wind Definitions (data) dialog** on page 3038
- **ASCE 7 Wind Load dialog box** on page 3039
- **M. To add an ASCE 7 wind load definition** on page 836

**AD.2005.1.2 Generation of Snow Load Per SEI-ASCE 7-02**

STAAD.Pro is now capable of generating snow loading on a structure in accordance with the provisions of the ASCE-7-02 code. The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

There are two parts to the command specification for snow load generation.
Part I

In this step, we define the parameters. In the General | Load page (from the vertical tabs on the left-hand side), go to the Load dialog box on the right and select Snow Definition as shown in the next figure.

Click on the Add button to instantiate the Add New: Snow Definition dialog box shown below. The Type No. is an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model.

The fields in the above dialog box are:
**Ground Snow Load**  The pressure or, weight per unit area, to be used for the calculation of the design snow load. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of SEI/ASCE 7-02.

**Exposure Factor**  Exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.

**Thermal Factor**  Thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.

**Importance Factor**  Importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

Click on the Add button within the dialog box and then click on Close.

The newly created data will appear underneath Snow Definition in the Load dialog box as shown below.

The members for which the snow load has to be generated have to be clustered together into FLOOR GROUPs. That is because, as we saw at the beginning of this section, the load generation is based on panel areas, not the exposed width of individual members. For details on creation of groups, refer to section 5.16 of the STAAD.Pro Technical Reference manual, and AD.2004.11 of the Software Release report for STAAD.Pro 2004.

**Part II**

Create a new primary load case. After the load case number and load title have been specified, click on Add.

Select it in the Load dialog box and click on the Add button to bring up the Add New: Load Items dialog box as shown below.
Select **Snow load** from the available list of load types. Various options corresponding to data required for snow load generation will appear on the right hand side, as shown in the figure above. The options are explained as follows:

**Floor Group**  
Select the floor group on which the snow load is to be applied.

**Condition**  
Specify whether the load is "" or "Unbalanced". These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.

**Define Snow Type**  
Select the snow load type number. This is the number specified against the **Type No.** field while creating the snow load definition in Part I described earlier in this section.

**Roof Type**  
Specify the roof type from the available choices:

- Default (if the roof type is not Mono, Hipped or Gable, it is referred to as Default)
- Mono (see mono-sloped roof shown in figure 6-6 of the code)
- Hipped (see figures 6-3 and 6-6 of the code)
- Gable (see figures 6-3 and 6-6 of the code)

These choices are described in section ... of ASCE 7-02.

**Roof Obstruction**  
Specify whether the roof is "obstructed" or "unobstructed". This also is a term described in section ... of ASCE 7-02.

**Roof Slope Factor**  
For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.

Click on **Add** to complete the data input for Part II. The snow load command will appear underneath the snow load case in the **Load** dialog box as shown below.
Related Links

- Add New Snow Definition dialog (on page 3043)
- TR.31.5 Definition of Snow Load (on page 2641)
- M. To add an ASCE 7-02 snow load (on page 851)
- Snow Load tab (on page 3021)
- TR.32.13 Generation of Snow Loads (on page 2785)
- M. To add an ASCE 7-02 snow load (on page 851)

AD.2005.1.3 Wall-slab Interface Considerations in Finite Element Meshing

In the plate element mesh generation for a panel type of entity like a wall or slab, STAAD.Pro now provides a facility for the consideration of boundary conditions at the interface of the panel and any other panel on whose surface one of its edge lies.

To illustrate the problem that one faces if the facility did not exist, consider the wall and slab shown in the figure below:

The Parametric Models facility may be used to mesh the slab, and use the wall boundary as a density line, resulting in the following mesh for the slab.
If the wall is to be meshed subsequently, the nodes along its common boundary with the slab, as shown in the above figure, must automatically be “control points” in the wall meshing process. There was no simple way to do this until now.

**Related Links**
- *M. To define a slab/wall connection* (on page 660)

**AD.2005.1.4 Enhancements to Renumbering of Entities**

The capabilities of the program in renumbering entities such as joints, members, plates, etc. have been enhanced. In the past, only a limited amount of control was available in the manner in which renumbering was to be done. The current enhancements enable a user to set multiple criteria and assign them an order of priority.

**Description**

We will use example problem 14 in the US examples folder to illustrate this. We will renumber the columns of one frame of this multi-story building. If you do not wish to modify the file supplied with the program, you may make a copy of the file before you start this exercise.

After opening the model, go to Select > By Range > XY, and set the minimum and maximum Z coordinate values to -1 ft and 1 ft respectively.
The members at the rear of the structure will be highlighted. Select View > New View > Display the view in the active window > .

Switch on the beam numbers and we get the following.
Our goal is to renumber the columns in such a manner that we will have the following once the renumbering is complete.
To accomplish that, do the following:

Select **Select > Beams Parallel to > Y**. The columns will be highlighted. Select **Geometry > Renumber > Members**.
What's New?
STAAD.Pro 2005 Release Report

The following message will be displayed.

Choose Yes, and the Renumber dialog box will appear.

It is apparent from the earlier figure that if we set the renumbering criteria to be based on X coordinate first and Y coordinate next, the program will internally create the following sequence for renumbering:

301, 317, 333, 302, 318, 334, 303, 319, 335, 304, 320, 336

So, the settings for the dialog box should be as follows:

Start numbering from = 1001
Select Sort Criteria = X Coordinate (Ascending), Y Coordinate (Ascending)

Click on Accept.
A message indicating the successful completion of the operation will appear.

Subsequently, the following view will appear on the drawing.

In past versions, 4 separate renumbering operations, each involving one line of columns, would have been required to achieve the same thing.

It is also worth noting that if the user wishes to renumber all the columns of all the frames in sequence, the Z coordinate can be included as a third criteria for sorting.

**Notes**

1. The order of importance of the sort criteria can be changed by clicking on the Up and the Down arrow keys as shown below.
Use these keys to change the order of the sort criteria

2. For the individual criteria, if you wish to change the order from Ascending to Descending or vice versa, double click on the item in the list as shown. For example, if you want the extreme right columns to have a lower number than the extreme left, set the criteria for X coordinate to Descending.
   Double-click on the individual items to change the order from Ascending to Descending and vice-versa.

3. If you want the entity to be numbered in the descending order instead of ascending order (as in 99, 98, 97, 96, ..., etc.), choose the appropriate button shown in the next figure.
   Click here to set the numbering sequence to Ascending or Descending.

Related Links
- Renumber dialog (on page 2904)
- M. To renumber selected beams (on page 649)

AD.2005.1.5 Property Calculator for User Table General Sections

The user table section type called General has been enhanced in the following manner:
a. The shape of the section can be defined in terms of coordinates of the corner points of its cross section outline.

b. A section property calculator built into that dialog box enables the values to be calculated instantaneously. Any values the user wishes to override can be changed by typing over them.

c. Points can be defined at which stresses are to be reported in the Beam-Stresses page of the post-processing mode.

Description

Select **Tools > Set Current Input Units**, and choose the length unit in which to specify the dimensions of the cross section. Let us choose inches for this exercise.

Next, select **Tools > Create User Table**. In the **Create User Provided Table** dialog box, click on **New Table**.

Select **General** for the **Select Section Type**. Click on **OK**.

The table number will appear as 1. Click on **Add New Property**. The following dialog box will appear.
Let us say that we would like to define a cross section as shown below.
Assuming the top left corner of the cross section to be (0,0) for (Y,Z), Z being along left-right and Y along top-bottom, specify the data as shown in the next figure. The sequence of steps is circled in the next figure. Once the points are specified, click on Compute Section Properties and observe the values being computed and filled in the relevant boxes (step 4 in the next diagram)

Steps
1. Select Define Profile Polygon
2. Type TWOLEGGEDTEE in the Section Name field.
3. Type the vertex coordinates in the table.
4. Click Compute Section Properties.

You will observe that when we clicked on Compute Section Properties, the Y and Z coordinates of the corner points that we typed earlier will be replaced by values that are measured from the center of gravity of the section.

If you wish to specify the locations on the cross section where stresses are desired, they must be provided too. Note that the coordinate locations for that box should be based not on the arbitrary datum point we started out with initially, but on the basis of the center of gravity of the section.
These are points where stress reports are desired in the **Beam | Stresses** page in the post-processing mode. The coordinates of these points are measured from the CG of the section and not from an arbitrary point.

Once the section is defined, it may be assigned from the General-Properties page in the conventional way. Some of those steps are shown in the next 2 figures.
For the case of the example above, the following command syntax will appear in the STAAD input file.

START USER TABLE
TABLE 1
UNIT INCHES KIP
GENERAL
TWOLEGGEDTEE
152 16 20 0 3395.93 4520.67 638.828 452.067 586.57 83.3308 -
46.3944 625.529 625.529 95637 .5 0
PROFILE POINTS
0 0 0 -4 3.5 -4 3.5 -16 6.5 -16 6.5 -4 13.5 -4 13.5 -16 16.5 -16 16.5 -4 20 -4 20 0
STRESS LOCATIONS
-10 5.78947 10 5.78947 -6.5 -10.2105 6.5 10.2105
END

Related Links
• M. To create a general section (on page 735)

AD.2005.1.6 Stretch Members

This new facility allows one to increase the length of a member in various ways. The benefit it offers is that a member may be extended even if one does not readily know the coordinates of its ends joints in its stretched condition, something a user may appreciate in the case of members whose axis lie at an inclination to the global planes.

Description
Select the member which is to be extended. In the Geometry menu at the top of the screen, the option called Stretch Selected Member(s) becomes active.
Click on it and the following dialog box will appear.

The various options of the dialog box are explained below.

Select member(s)  Multiple members can be selected simultaneously for stretching. However, whether they will get stretched or not depends upon the type of method used in stretching as described later. If you wish to remove one or more members from the list, uncheck the corresponding boxes.
Uncheck those members which you do not wish to stretch.

The methods available for stretching are:

**To a point**
Specify the coordinates in current length units of the point to which one of the ends of the member are to be moved. The point must lie on the axis of the member being stretched, or else, the stretching will not be performed. The program automatically determines which of the two ends of the member is to be moved. So, this method involves

i. determining if the point lies on the axis of the member(s) being stretched.
ii. determining which end to move for the member(s) which satisfy criteria (i)
iii. replacing the corresponding nodal coordinates with those of the desired point.

**Through a distance**
In this method, the user has to merely specify the distance by which the start or end node must be stretched.

Choose the end which should be stretched. Specify the distance by which to stretch.

In this method, all members selected for this operation will see the change in length. The next diagram illustrates this. If this causes member to cross each other, the user must create the intersection point using the Geometry – Intersect Selected Members tool as the program does not automatically create it in the Stretch operation.
To an existing node
This is very similar to the To a point method described earlier. The only difference is that instead of explicitly specifying the coordinates of the desired point, that point is already available for identification through its node number.

To an existing member
If 2 members are oriented in such a manner that the local X-axis of one of those members can potentially intersect a second member within the span of that member, this method may be used to stretch the first member to meet the second member. In the next figure, member 42 may be stretched so it meets member 73. The second member will be automatically split up at the intersection point into two segments.

The dialog box settings required to achieve this are as follows:
Notes

In the **To an existing node** method, a graphical tool is available for selecting the node from the drawing as shown in the next figure.

In the **To an existing member** method, a graphical tool is available for selecting the member from the drawing as shown in the next figure.

In the **To an existing node** method, use this tool graphically select an existing node from the model.

In the **To an existing member** method, use this tool graphically select an existing node from the model.

The **Details** button may be switched on to view the internal details of the stretch operation.
AD.2005.2 Features Affecting the Post-Processor

The additions and enhancements to the post-processor section of the program are explained in the following pages.

AD.2005.2.1 Floor Vibration Analysis

The adequacy of a floor system from the standpoint of its vibration serviceability due to human activity, specifically walking excitation, can now be assessed using STAAD.Pro. The procedures of Chapters 3 and 4 of the AISC Steel Design Guide Series No. 11 - Floor Vibrations due to Human Activity - have been implemented.

Tutorial Problem

A composite deck is system composed of a concrete slab lying over a steel deck with or without ribs. The steel deck in turn is supported by steel beams or joists and they span the distance between girders and are supported by those girders. The slab may or may not be connected to the joists by shear studs.

To model this system in STAAD, one has to go to Geometry > Composite Deck from the left side of the screen as shown in the next figure:
The **Composite Deck** dialog opens on the right side of the screen as shown in the next figure.
When you click on the **Create New Deck** tab, you will see the mouse cursor change to look like an icon of a colored composite deck. To define the periphery of the composite deck, click the corner nodes in clockwise or counter-clockwise sequence using this mouse cursor. The last click must be on the starting node to close the periphery. In the figure shown, the sequence used is A-B-C-D-A.

The name of the composite deck will appear in the **Composite Deck** dialog. Click on that name and then dialog box will display several additional contents as shown in the next figure.
In the main view, the composite deck will appear in hatched lines. An arrow mark will indicate the direction along which the composite deck spans. This arrow will appear by default. To change the span direction, select two beams whose X axis is perpendicular to the intended span direction, and click on the tab **Create Direction**.

Define the concrete properties, rib properties and connectivity details in the appropriate fields of the **Composite Deck** dialog. To save the properties, click on the tab **Update Deck Property** (see previous figure). A sample data set is shown in the next figure.

**Command in the STAAD Input File**

When the data is specified in the dialog boxes as we saw earlier, it is also simultaneously stored in the STAAD input file in the appropriate command syntax. For the data we specified previously, the corresponding editor input will be as follows:

```plaintext
START DECK DEFINITION
  _DECK FLOOR1
  PERIPHERY 1 TO 8
  DIRECTION -1.000000 0.000000 0.000000
  COMPOSITE 10 9 4 8
```
Floor Vibration Report

In order to obtain the report, the finished model must be successfully analyzed. Go to the post-processing mode. Select Report > Floor Vibration Report as shown in the next figure.

The **Floor Vibration Output** dialog as shown in the next figure will appear. Select the deck name, the load case, and click on Check to see the report.

1. Select the composite deck from the drop-down list.
2. Select the load case.
3. Click **Check**.
What's New?
STAAD.Pro 2005 Release Report

An output similar to the one shown below will appear.

The terms displayed in the above box have been explained earlier.

Example STAAD Input File

```
STAAD SPACE
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 20 0 0; 4 30 0 0; 5 0 0 35; 6 10 0 35; 7 20 0 35;
8 30 0 35; 9 0 -15 0; 10 30 -15 0; 11 0 -15 35; 12 30 -15 35; 13 0 15 0;
14 30 15 0; 15 0 15 35; 16 30 15 35; 17 10 15 0; 18 20 15 0; 19 20 15 35;
20 10 15 35;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 8; 5 8 7; 6 7 6; 7 6 5; 8 5 1; 9 2 6; 10 3 7; 11 1 9;
12 4 10; 13 5 11; 14 8 12; 15 1 13; 16 4 14; 17 5 15; 18 8 16; 19 13 17;
20 17 18; 21 18 14; 22 14 16; 23 16 19; 24 19 20; 25 20 15; 26 15 13; 27 17
20;
28 18 19;
DEFINE MATERIAL START
ISOTROPIC STEEL
```
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-006
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
4 8 TO 18 22 26 TO 28 TABLE ST W18X35
1 TO 3 5 TO 7 19 TO 21 23 TO 25 TABLE ST W21X50
START DECK DEFINITION
_DECK C2
PERIPHERY 1 TO 8
DIRECTION -1.000000 0.000000 0.000000
COMPOSITE 10 9 4 8
OUTER 1 4 8 5
DIA 0.000000
HGT 0.000
CT 0.271
FC 576.000
RBW 2.000
RBH 0.167
SHR 0
VENDOR NONE
CD 0.110
CMP 2.0
CW 10.000000 MEMB 10
CW 10.000000 MEMB 9
CW 5.000000 MEMB 4
CW 5.000000 MEMB 8
_DECK C3
PERIPHERY 19 TO 26
DIRECTION -1.000000 0.000000 0.000000
COMPOSITE 28 27 22 26
OUTER 13 14 16 15
DIA 0.000000
HGT 0.000
CT 0.800
FC 476.000
RBW 0.500
RBH 0.500
SHR 0
VENDOR NONE
CD 0.150
CMP 2.0
CW 10.000000 MEMB 28
CW 10.000000 MEMB 27
CW 5.000000 MEMB 22
CW 5.000000 MEMB 26
END DECK DEFINITION
CONSTANTS
MATERIAL STEEL MEMB 1 TO 28
SUPPORTS
9 TO 12 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
SELFWEIGHT Y -1
UNIT FEET POUND
ONEDAY LOAD
YRANGE 0 0 ONE -57 GY
AD.2005.3 Features Affecting Analysis and Design

Some new features have been added and existing features have been modified in the analysis and design part of
the program. They are explained in the following pages.

AD.2005.3.1 Designing I-beams w Web Openings per AISC ASD

Design of steel members with web openings per AISC Steel Design Guide 2 - ASD specifications is now available
in STAAD. The facility is available for members whose yield strength is 65 ksi or less.

Note: In the current implementation, the web openings are given consideration only during the design phase.
The reduction in section properties caused by the presence of the openings is not considered automatically
during the analysis phase. Hence, the analysis is performed as if the full section properties are effective for such
members.

During the design process, the program first determines the utilization ratio (U.R.) at the location of the opening
as though it is an unreinforced opening. If the U.R. is less than 1.0, the member is presumed to have passed the
requirements at that location. If the U.R. exceeds 1.0, then it determines the U.R. as though it is a reinforced
opening. If it fails this too, the cause of the failure along with the associated numerical values is reported.
T.1 – Steel Portal Frame

This chapter provides a step-by-step tutorial for creating a 2D portal frame using STAAD.Pro.

![Portal Frame model](image)

**Figure 9: Portal Frame model**

**T.1 Methods of creating the model**

There are three methods of creating the structure data:

1. Using the STAAD.Pro Physical Modeler interface (on page 377),
2. Using the traditional STAAD.Pro analytical modeling interface (on page 385),
3. Using the command file (on page 428)

The physical model is used to draw structural elements as they are physically constructed. The program will then decompose this into an analytical modeling which is passed to the STAAD.Pro analysis and design engine when you run your model.

The analytical model is a finite element model of the structure which is typically processed directly by the analysis and design engine.
The command input file is a text file which contains the data for your structural model. This file consists of simple English language-like commands. This file is created for you behind the scenes when you model your structure using either the physical modeler interface or the analytical model interface.

The physical model will be linked to the command input file so any changes to your physical model must be made using the physical modeler interface. This ensures data integrity when the program decomposes the physical model. You can “break” this link to make changes to the underlying analytical model directly, either using the STAAD.Pro analytical modeling interface or using the command input file editor.

You can switch back and forth between the analytical modeling interface and the command input file editor for models created by either of those methods.

All three methods are described in this tutorial, with a detailed focus on the STAAD.Pro Physical Modeler interface.

**T.1 Description of the tutorial problem**

The structure for this project is a single bay, single story steel portal frame that will be analyzed and designed.

![Figure 10: Portal Frame model](image)

**Table 21: Basic Data for the Structure**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member properties</td>
<td>Members 1 &amp; 3 : W12X35</td>
</tr>
<tr>
<td></td>
<td>Member 2 : W14X34</td>
</tr>
<tr>
<td>Material Constants</td>
<td>Modulus of Elasticity : 29,000 ksi</td>
</tr>
<tr>
<td></td>
<td>Poisson’s Ratio : 0.30</td>
</tr>
</tbody>
</table>
### Attribute | Data
--- | ---
Member Offsets | 6.0 inches along global X for member 2 at both ends
Supports | Node 1: Fixed  
Node 4: Pinned
Loads | Load case 1: Dead + Live  
Beam 2: 2.5 kips/ft downward along global Y  
Load case 2: Wind From Left  
10 kips point force at Node 2  
Load case 3: 75 Percent of (DL+LL+WL)  
Load Combination - L1 X 0.75 + L2 X 0.75
Analysis Type | Linear Elastic (PERFORM)
Steel Design | Consider load cases 1 and 3 only.  
Parameters: Unsupported length of compression flange for bending: 10 ft for members 2 and 3, 15 ft for member 1.  
Steel Yield Stress: 50 ksi  
Perform member selection for members 2 and 3

#### T.1 Creating a new structure

On the Start page **New** tab, you will provide some initial data necessary for building the model.

1. On the Start page, select **New**.

   The **New** page opens to the **Model Info** tab.

2. Type **PORTAL** in the **File Name** field.
3. Specify a **Location** where the STAAD input file will be located on your computer or network.

   You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.

4. Select **Physical** for the **Type** of model.

   This option selects the modeling method you want to use:

   - **Analytical** For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.
   - **Physical** For creating a model using the STAAD.Pro Physical Modeler interface.
   - **Building** For creating a building model structure using the Building workflow.

5. Select **English** as the system of **Units**.

   **Tip:** The units can be changed later if necessary, at any stage of the model creation.

6. (Optional) Select the Job Info tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.

7. Click **Create**.

The STAAD.Pro modeling environment opens and your model file is then opened in the STAAD.Pro Physical Modeler.

Alternatively, if you want to use either the analytical modeling workflow or the command input file editor to create the model, choose **Analytical**, click **Create**, and then proceed to either "[T.1 Creating the model using the analytical user interface](#)" (on page 385) " or "[T.1 Creating the model using the command file](#)" (on page 428)," respectively.
T.1 Creating the Model using the Physical Modeler

You are now ready to start building the model geometry. The steps for doing this are described in the following sections.

**Tip**: Refer to the STAAD.Pro Physical Modeler Application Window Layout for reference on the application window.

### T.1 Generating the model geometry

1. On the **Model** ribbon tab, select the **Grid** tool in the **Create** group.

   ![Create Grid dialog](image)

   The **Create Grid** dialog opens.

2. Supply the grid details:
   a. Type **Grid 1** in the **Name** field.
   b. Leave the **Plane** selection as **XY** and the **Creation Method** as **By Spacing**.
   c. Type 20 in the **Number of spaces in X** and 1 ft in the **Grid spacing X** fields.
   d. Type 15 in the **Number of spaces in Y** and 1 ft in the **Grid spacing Y** fields.
   e. Click **OK**.

   The grid is created.

   You can control the visibility of individual grids by selecting the **Grids** tool on the **Spreadsheet** ribbon tab, where you can also change the settings for each grid. You can have as many grids as necessary displayed at one time.

3. On the **Model** ribbon tab, select the **Member** tool in the **Create** group.

   ![Member tool highlighted](image)

   The tool is highlighted to show it is active. A yellow circle highlights the grid intersection the cursor “snaps” to when you hover in the view area.

4. Click on the lower, right corner of the grid (i.e., the origin at 0,0,0).

   A dashed line is “rubber banded” from this point to your cursor.

5. Click on the following two points around the edge to draw two members:
   
   (0,15,0)
   (20,15,0)

6. Double-click on the final point at (20,0,0).

   This instructs the program you are finished drawing connected members.

   **Tip**: The tool remains active until so you can continue drawing a new set of connected members elsewhere if necessary.

7. Select the **Member** tool again to make it inactive.

8. On the **View** ribbon tab, select the **Grids** tool in the **Reference** group.
The tool is no longer highlighted and all grids are hidden in the display.

9. Switch on the node and beam labels:
   a. On the View ribbon tab, select the drop-down list below the Numbering tool in the Model group.

   - Select Node and Member on this list.
   - They are both marked with a check.
   - Select the Numbering tool.

You have now created the nodes and members for this model.

You can review the geometry of each by selecting the Nodes tool in the Nodes group and Members tool in the Members group on the Spreadsheet ribbon tab, respectively.

**T.1 Specifying member properties**

*Tip:* In STAAD.Pro Physical Modeler, you will select a model object and then assign properties, loads, etc. to that object. This fundamental workflow is import to understand as it applies to essentially all actions once you have drawn your model objects.

1. Select the two column members (M1 and M3):
   a. Click anywhere in the empty area in the view window.
      All model objects (members and nodes) are deselected.
   b. Click-and-drag a line horizontally from right to left across the columns to select them.
      This method (i.e., right to left) will select any model object that your drag window crosses.
Note: The Member ribbon tab reappears any time you have one or more members in your current selection.

2. On the Member ribbon tab, select the Section tool in the Assign properties group.

The Assign Section dialog opens.

3. Specify the section data for the columns:
   a. Leave all the defaults:
      - Type: Standard,
      - Country: United States,
      - Material: STEEL,
      - Category: Hot Rolled,
      - Specification: AISC,
      - Version: 14 Edition,
      - Manufacturer: Generic
   b. Select W from the Table drop-down list.
      The list of all section names in this table are populated.
   c. Scroll to and select W12X35 in the list.
   d. Click OK.
   The section is assigned to the columns.

4. Click on the beam member.
   It is selected and all other members are deselected.

5. Repeat steps 2 and 3 except to assign a W14X34 to the beam (M2).

6. Select all three members by either:
   - click-and-drag a window around them
   - press <Ctrl+A>
   - or
   - on the Data ribbon tab, select the Select all tool in the Model group

   Tip: There are numerous ways in which to select all the beams in a model. These are some examples.

7. On the Member ribbon tab, select the Material tool in the Assign properties group.

The Assign Material dialog opens.

8. Specify the material data for the members:
a. Leave the initial defaults:
   
   Source: Catalog
   Type: Standard
   Country: United States
b. Select ASTM_STEEL from the Specification drop-down list.
c. Select A992 from the Name list.
d. Click OK

The material is assigned to the members.

T.1 Specifying member offsets

1. Click the beam member (M2) to select it.
2. On the Member ribbon tab, select the End Offset tool in the Edit group.

   The Modify Member End Offsets dialog opens.
3. Specify the start (in this case, left) node offset:
   a. Leave the Direction as Local.

   Tip: Given the orientation of this beam, the local 1 and global X axis are parallel, but it's best practice to use local coordinates when the offset is taken with respect to the local member's ends.
   
   b. Type 6 in in the x direction.
   
   Local axis 1, 2, and 3 correspond to the member's localized X, Y, and Z axis.

   Tip: Notice that even though the default input units are in feet, you are able to type different units and the program converts to the input units for you.

4. Specify the end (in this case, right) node offset:
   a. Leave the Direction as Local.
   b. Type -6 in in the x direction.

   Note: The negative sign for the offset value at the end.

5. Click OK

The end offsets are drawn as red lines from the nodes in their indicated direction.

T.1 Specifying supports

1. Click on the lower-left node (N1) to select it.

   Any other selected items are removed from the current selection set. The node is selected. The node label appears if you have the labeling turned on.

2. On the Node ribbon tab, select the Fixed tool in the Supports group.

   A fixed nodal support type is assigned to this node.

3. Select the lower-right node (N4).
4. On the Node ribbon tab, select the Pinned tool in the Supports group.
A pinned nodal support type is assigned to this node.

**Tip:** You can use the Custom tool in the Supports group to generate any restraint or spring nodal boundary condition necessary.

You can review the nodal boundary conditions by selecting either the Supports tool or the Springs tool in the Nodes group on the Spreadsheet ribbon tab.

### T.1 Viewing the model in 3D

1. Select all model objects by either:
   - Right-click in the view area and select **Select All** from the pop-up menu
   - Click-and-drag a window area around all model objects
   - Press `<Ctrl+A>`
2. On the View ribbon tab, select the 3D Rending tool in the Model group.

   The tool is highlighted to indicate it is active. The view displays the rendered model using the member’s shapes, materials, and properties such as end offsets.

   You can continue to model and work with this view option turned on. However, it is typically clearer to work with line drawings. Select the 3D Rendering tool again to deactivate it.

### T.1 Specifying loads

The STAAD.Pro Physical Modeler creates a default Load Group 1 with the no specified load type. You will add the dead and live loads to this load group and then create a new load group for the wind load.

1. On the Spreadsheet ribbon tab, select the Load cases tool in the Reference group.

   The Load Cases spreadsheet opens.

2. For Load case 1, type Dead + Live in the Name cell to rename the load case.
3. Click the beam member (M2) to select it.
4. On the Member ribbon tab, select the Distributed tool in the Loads group.

The Add Member Distributed Load dialog opens.

5. Specify the load:
   - Leave the Load Group as LC: Dead + Live and the Load Type as Uniform.
   a. Select Global Y as the Direction.
   b. Type -2.5 kip/ft in the Magnitude field.
   c. Click OK.

   The load is applied to the beam.

6. On the Model ribbon tab, select the Load Case tool in the Loads group.

The Add Load Case dialog opens.

7. Specify the wind load case details:
   a. Type Wind From Left in the Name field.
   b. Select Wind for the Type.
   - Leave the other items for Self-weight data empty (default).
   c. Click OK.

   A new load case is added to the Load Cases spreadsheet and the load is automatically selected in the program window status bar as the current load.

   **Note:** When load items are created they are added to the current load case or load group by default.

8. Select the upper-left node (N2).
The **Add Nodal Load** dialog opens.

10. Specify the lateral wind load:
    a. Leave the **Load group** selection as **LC: Wind from left**.
    b. Type 10 kip in the **Fx** field.
    c. Click **OK**.

### T.1 Creating a load combination

1. On the **Model** ribbon tab, select the **Load Combination** tool in the **Loads** group.

   ![Image]

   The **Add Load Combination** dialog opens.

2. Type **75 Percent of [DL+LL+WL]** in the **Name** field.

3. Select **Linear** as the load combination **Type**.

4. Click in the **Load case filter** field and then check the **(Select All)** option.

5. Assign the load case factors:
    a. Click in the **Factors** field.
       The load case factors table opens. All the load cases in the model are included from the previous step.
    b. For load case 1 (the first row in the table), click in the **Factor** cell and type **0.75**.
    c. Repeat step 5a to enter a load case factor of 0.75 for the wind load (load case 2).
    d. Click **OK**.
6. Click OK.

T.1 Generate the analysis model

You have now completed the modeling portion of this tutorial. You will now send the model data back to STAAD.Pro where you will create load combinations, add output commands, select analysis criteria, and specify design parameters.

For this tutorial, you will use the default options for analytical model generation. However, for advanced models you can review and change those options in the Configuration dialog.

1. Select Save to save the physical model.
2. Either:
   - On the Model ribbon tab, select the Return to Analytical Modeling tool in the STAAD.Pro group
or

On the **File** ribbon tab, select **Create analysis model** in the backstage tabs.

The Return to Analytical Modeling dialog opens and displays the progress of the analytical model being generated.

3. Click **OK**.

If you have not previously associated this model with a CONNECT Project, you are asked to do so. This is will not be necessary for this tutorial.

4. Click **Cancel** in the **Assign Project** dialog.

The STAAD.Pro Physical Modeler window closes and the model is loaded into the STAAD.Pro analytical modeling interface.

**T.1 Creating the model using the analytical user interface**

The following procedures describe how to use the traditional STAAD.Pro analytical user interface to build the model. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

**Tip:** Refer to Application Window Layout (on page 50) for reference on the application window.

**Note:** If you completed the model using the **Physical Modeling** workflow, then instead proceed to T.1 Analysis and Design (on page 412).

**T.1 Generating the model geometry**

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc.

The STAAD input file commands generated are:

```
JOINT COORDINATES
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
```
When starting in the Analytical Modeling workflow, a grid is initially displayed in the main view window. This grid is controlled by the **Snap Node/Beam** dialog. The directions of the global axes (X, Y, Z) are represented in the icon in the lower left hand corner of the drawing area.

![Figure 11: The STAAD.Pro window with the Snap Node/Beam dialog open](image)

**Note:** The **Snap Node/Beam** dialog opens on the right side of the screen. You can reopen this dialog by selecting the **Snap Grid Beam** tool in the **Structure** group on the **Geometry** ribbon tab.

1. On the **Snap Node/Beam** dialog, click **Create**. A dialog opens which will enable us to set up a grid.

   Within this dialog, there is a drop-down list from which you can select Linear, Radial, or Irregular form of grid lines.

   - **Linear** used to place the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board

   - **Radial** used to place construction lines to appear in a spider-web style, which makes it easy to create circular type models where members are modeled as piece-wise linear straight line segments

   - **Irregular** used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane

2. Select **Linear**, which is the **Default Grid**. In this structure, the segment consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane. Leave X-Y as the **Plane** of the grid. The size of the model that can be drawn at any time is controlled by the number of **Construction Lines** to the left and right of the origin of axes, and the **Spacing** between adjacent construction lines.

3. Type a **Name of Grid 1**.

4. Type 20 as the number of lines to the **Right** of the origin along X and 15 above the origin along Y
Leave the default spacing of 1 feet between lines along both X and Y.

![Figure 12:](image)

5. Click **OK**.

6. In the **Snap Node/Beam** grids list, check the new **Grid 1** option.

   You can create any number of grids. By providing a name, each new grid can be identified for future reference.

   **Tip:** Please note that these settings are only used to generated construction lines. These construction lines enable you to easily draw the structure but do not restrict our overall model to those limits.

   **Tip:** To change the settings of this grid, select the name in the **Snap Node/Beam** dialog and then click **Edit**.

The check by **Default Grid** is automatically cleared. STAAD.Pro will only display a single grid at a time.
7. In the View window, click at the origin (0, 0) to create the first node.

   A line is "rubber-banded" between this node and the mouse pointer, which previews the member to placed with the next mouse click.

   The **Snap Node/Beam** feature is active by default.
8. Click on the following points to create nodes and automatically join successive nodes by beam members.

(0, 15)  
(20, 15)  
(20, 0)  

When steps 1 through 5 are completed, the structure will be displayed in the drawing area as shown below.

Figure 14:
9. Click **Close** in the **Snap Node/Beam** dialog. The grid is hidden.

![Figure 15: The portal frame members drawn](image)

It is important to save your work often. This helps to avoid loss of data and protect your investment of time and effort against power interruptions, system problems, or other unforeseen events.

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL + S>`.

### T.1 Switching on node and beam labels

Node and beam labels are a way of identifying the entities in the View window.

1. Right click anywhere in the view window and select **Labels** from the pop-up menu.
   - The **Diagrams** dialog opens to the **Labels** tab.
2. Set the **Node Numbers** and **Beam Numbers** on and then click **OK**.
Figure 16: Select the Node and Beam numbers

**Tip:** Alternatively, you can press `<Shift+N>` and then `<Shift+B>` to quickly toggle the same labels. The letters in parenthesis on the *Labels* tab indicate these `<Shift>` key shortcuts.

**Tip:** You can change the font of the node/beam labels by selecting the File ribbon tab and then selecting **Settings > Display Options**. Select the appropriate tab (Node Labels / Beam labels) from the Options dialog.
T.1 Specifying member properties

In this procedure, you will assign cross section properties to the beams and columns. The STAAD input file commands generated are:

```
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X35
2 TABLE ST W12X34
```

1. Select the **Properties** page in the Analytical Modeling page control bar.

   ![Properties - Whole Structure dialog](image)

   *Figure 17:*

2. Click **Section Database**.

   The **Section Profile Tables** dialog opens.
3. Select the **W Shape** tab under the **American** option.
   The property type we wish to create is the W shape from the AISC table.

4. Choose **W12X35** as the beam size and then select the **ST** option as the section type.
   **Tip:** The **Material** check box is set. Leave this set and the selection set to **Steel** as it will be used to assign the built-in steel material properties when this section is assigned.

5. Click **Add**.
   **Tip:** Detailed explanation of the terms such as ST, T, CM, TC, BC, etc. is available in Section 5 of the STAAD Technical Reference Manual.

The W12X35 section is added to the Sections list in the **Properties - Whole Structure** dialog.
6. To create the second member property (ST W14X34), select the **W14X34** shape and click **Add**.

7. Click **Close** in the **Section Profile Tables** dialog.

8. Assign these properties with selected members in our model:
   a. Select the first property reference in the **Properties** dialog (W12X35).
   b. Select the **Use Cursor to Assign** option in the Assignment Method group.
   c. Click **Assign**.
   
   The mouse pointer changes to 
   
   d. Click on members 1 and 3.
   e. To stop assigning properties, either:
      
      click **Assigning**
      
      or
      
      press the `<Esc>` key.

9. Repeat step 8 except to assign the second property reference (W14X34) to member 2.

After the properties are assigned to the respective members, the member labels will indicate the section reference numbers.

![Portal.std - Whole Structure](image)

*Figure 19:*

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL+S>`.

---

**T.1 Specifying material definition**

When selecting the steel sections (on page 392), you kept the **Material** option checked. Consequently, the material definitions were assigned to the members along with the sections.
The STAAD input file commands generated are:

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
... 
CONSTANTS
MATERIAL STEEL ALL
```

T.1 Changing the input units of length

For specifying member offset values and for some material definition values, as a matter of convenience, use length units of inches instead of feet.

The STAAD input file commands generated are:

```
UNIT INCHES KIP
```

1. On the Geometry ribbon tab, select the Input Units tool in the Structure group.

The Set Input Units pop-up opens.

2. Select English as the Input Units.
3. Set the Length input units to Inch.
4. Click Apply.
T.1 Specifying member offsets

Member 2 (the beam) actually spans only the clear distance between the column faces rather than the center to center distance. This is modeled by specifying offsets.

Member 2 is offset at the start joint by 6 inches in the global X direction (and 0.0 and 0.0 in Y and Z directions). The same member is offset by negative 6.0 inches at its end joint.

The STAAD input file commands generated are:

```
MEMBER OFFSET
  2 START  6.0  0.0  0.0
  2 END    -6.0  0.0  0.0
```

1. Select member 2 by clicking on it in the view window.

   **Note:** If the beam cursor isn’t currently active, select the **Beam Cursor** tool in the **Selection** group on the **Geometry** ribbon tab.

   The selected member will be highlighted.

2. Select the **Specifications** page on the Analytical Modeling page bar. The **Specifications - Whole Structure** dialog opens. Member releases, offsets, and other beam specifications are defined here.

3. Click **Beam**.
The Member Specification dialog opens.

4. Specify the member start offset:
   a. Select the Offset tab.

   ![Member Specification dialog]

   b. Select Start as the Location.
   c. Type 6.0 (in.) in the X offset field.

   Leave the Direction as Global. It is convenient to define the offset at the start node in the X direction. Since the beam member is aligned with the global X axis, there is no difference between choosing a global or local frame of reference.

d. Click Assign.

The dialog closes and the offset specification is added to the start of member 2. This is displayed visually on the member as well as listed in the Specification list in the Specifications - Whole Structure dialog.

5. On the Specification ribbon tab, select the Beam > Offset tool in the Specifications group.

   ![Beam > Offset tool]

   The Member Specification dialog opens to the Offsets tab.

6. Specify the member end offset:
   a. Select End as the Location.
   b. Type -6.0 (in.) in the X offset field.
   c. Click Assign.

The dialog closes and the offset specification is added to the start of member 2.
Tip: Alternatively to steps 5 and 6, you can repeat steps 3 and 4, except for selecting the End option and providing a value of -6.0 for X. This procedure is used to demonstrate two methods to complete a similar task.

Figure 20: The model after both start and end offsets are assigned

Click anywhere in the drawing area to deselect the member.

Tip: Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL+S>.

T.1 Printing member information in the output file

It is useful to view the member information used by the program for analysis. You can add a report in the STAAD output file consisting of information about all the members including start and end joint numbers (incidences), member length, beta angle, and member end releases.

The STAAD input file commands generated are:

PRINT MEMBER INFORMATION ALL

1. Select all the beam members by one of the following methods:
   on the Select ribbon tab, select the All tool in the Beams group
   or
   click-and-drag a window around all members in the View window
   or
   press <Ctrl+A>

2. On the Analysis and Design ribbon tab, select the Pre-Analysis Commands tool in the Analysis Data group.
The Pre Analysis Print - Whole Structure dialog opens.
3. In the Pre Analysis Print - Whole Structure dialog, click Define Commands.
   The Analysis/Print Commands dialog opens.
4. Select the Member Information tab and then click Assign.

5. Click Close.

Click anywhere in the drawing area to deselect the member.

Tip: Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL +S>.

T.1 Specifying Supports

The boundary conditions of this problem call for restraining all degrees of freedom at node 1 (FIXED support) and a pinned type of restraint at node 4 (restrained against all translations, free for all rotations).

The STAAD input file commands generated are:

SUPPORTS
1 FIXED ; 4 PINNED
1. On the Select ribbon tab, select the Node Cursor tool in the Cursors group.

2. In the View window, click on nodes 1 to select it.

   Note: The Node Tools ribbon tab opens when nodes are selected.

3. On the Node Tools ribbon tab, select the Assign Supports tool in the Model dialog.

   The Supports - Whole Structure dialog opens.

4. In the Supports dialog, click Create.

   The Create Support dialog opens.
5. Select the **Fixed** tab (selected by default) and then click **Assign**.

   This creates a FIXED type of support at node 1 where all 6 degrees of freedom are restrained.

6. To create a PINNED support at node 4, repeat steps 2 through 5, except for selecting node 4 and selecting the **Pinned** tab in the Create Support dialog.
Figure 21: The portal frame with supports assigned

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL>+<S>`.

T.1 Viewing the model in 3D

You can review the model in 3D in the STAAD.Pro user interface.

Figure 22: The model rendered in three dimensions

1. Right-click in the view window and select **Structure Diagrams** from the pop-up menu.
The **Diagrams** dialog opens to the **Structure** tab.

The options under **3D Sections** control how the members are displayed.

- **None** displays the structure without displaying the cross-sectional properties of the members and elements.
- **Full Sections** displays the 3D cross-sections of members, depending on the member properties.
- **Sections Outline** displays only the outline of the cross-sections of members.

2. Select **Full Sections** and then click **OK**.
Tip: You can also change the color of the sections by clicking on the Section Outline color button in the Colors group.

Alternatively, you can quickly render the model in 3D in a new window either by:

- selecting the 3D Rendering tool in the Windows group on the View ribbon tab
- selecting View > 3D Rendering.

This method has no further settings for colors.

### T.1 Specifying Loads

Three load cases are to be created for this structure. Details of the individual cases are explained at the beginning of this tutorial.

The STAAD input file commands generated are:

```
UNIT FEET KIP
LOADING 1 DEAD + LIVE
MEMBER LOAD
  2 UNI GY -2.5
LOADING 2 WIND FROM LEFT
JOINT LOAD
  2 FX 10.
LOAD COMBINATION 3 75 PERCENT OF (DL+LL+WL)
  1 0.75 2 0.75
```

The creation and assignment of load cases involves the following steps:

1. Create the two basic load cases
2. Create a load combination as a third load case
3. Assign these loads to the respective members/nodes.

### T.1 Creating Load Cases 1 and 2

Load cases 1 and 2 are primary load cases. These load cases are used during analysis of the structure.

1. Change the length units to Feet by selecting the length unit in the application window status bar. See **T.1 Changing the input units of length** (on page 395).
2. On the Loading ribbon tab, select the Primary Load Case in the Loading Specifications group.

The Create New Primary Load Cases dialog opens.
3. Enter the properties for the first load case:
   a. Type DEAD + LIVE in the Title field.
      Leave the Number as the default value of 1.

      **Note:** The Loading Type list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program’s automatically generating load combinations in accordance with those codes. Note that there is a check box labeled Reducible per UBC/IBC. This feature is active only when the load case is assigned a Loading Type called Live when you create that load case.

      Since this tutorial does not use the automatic load combination generation feature, leave the Loading Type as None.

   b. Click Add.
      The load case appears under the Load Cases Details option in the Load & Definition dialog.

   c. Click Close.

4. On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group.
The **Create New Load Items** dialog opens with entries for adding load items to the current load case.

5. Define the uniform load for this load case:
   a. In the **Create New Load Items** dialog, select the **Uniform Force** tab under the **Member Load** item.
   b. Select **GY** as the **Direction**.
   c. Type **-2.5** in the **W1** field (magnitude of the uniform force).
      - The negative value indicates the load acts "down".
      - Leave the offset fields (d1 through d3) as zero.
   d. Click **Add**.
   e. Click **Close**.

6. Repeat steps 2 and 3 to create the second load case, except title the load case as **WIND FROM LEFT**.
   - Again, leave the **Loading Type** as **None**.
7. Set the wind load case as the current load case:
   a. On the **Loading** ribbon tab, click the down arrow on the **Load** tool in the **Display** group. The **Select load** pop-up dialog opens.

   ![Load & Definition dialog](image)

   b. Select the **2: Wind From Left** primary load case. Load items created using the ribbon tab tools are added to the current load case.

8. On the **Loading** ribbon tab, select the **Load Items** in the **Loading Specifications** group.

   ![Create New Load Items](image)

   The **Create New Load Items** dialog opens with entries for adding load items to the selected load case.

9. Define the wind load at the joint:
   a. In the **Create New Load Items** dialog, select the **Node** tab under the **Nodal Load** item.
   b. Type 10 in the **Fx** field (magnitude of the force along the global X direction).
   c. Click **Add**.
   d. Click **Close**.
T.1 Assigning load cases to members

Load item assignments are made using the **Load & Definition** dialog, which opens when most of the Loading tools are selected. If this dialog is not currently open, select the Loading page in the Analytical Modeling page control bar to re-open it.

You may notice that the load items in the Load & Definition dialog are listed with an icon. This indicates that these load items are not yet assigned to any model objects (i.e., beams or nodes).

1. In the **Load & Definition** dialog, select the **UNI GY -2.5 kip/ft** load item in the **1: DEAD + LIVE** load case.

![Load & Definition dialog](image)

2. Select the **Use Cursor to Assign** option in the **Assignment Method** group.
3. Click **Assign**.

   The mouse pointer changes to 

4. Click on member 2.

   The uniform load is drawn on member 2 and the load item icon changes to \(\text{uniform load}^{*}\) in the **Load & Definition** dialog.

5. Repeat steps 1 through 4, except to assign the nodal load item in the second load case (FX 10 kip, ft) to Node 2.

*Figure 23: Uniform load item assigned to member 2*
6. To stop assignment load items:
   
   click **Assigning**
   
   or
   
   Press the <Esc> key.

What about the load combinations? These do not need to be assigned to any model objects. As you may recall, these represent a factored combination of the *results* of the primary load cases and thus are not directly analyzed.

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing <CTRL +S>.

---

### T.1 Creating Load Case 3

In this procedure, you will add a load combination of the first two primary load cases.

If you started your model in STAAD.Pro Physical Modeler, first select the **Loading** page in the Analytical Modeling page control bar. The **Load & Definition** dialog opens on the right side of the program window.

Here you will see that the two load groups you created in STAAD.Pro Physical Modeler have each been added as primary load cases to the analytical model. The loads you added to the physical model are also transferred as load items in the respective load cases.

Load case 3 is a load combination. This means that the results from the primary load cases will be combined as directed by this load case definition.

1. On the **Loading** ribbon tab, select the **Combination Load Case** tool in the **Loading Specifications** group.

The **Create New Load Combinations** dialog opens.
2. Type 75 Percent of \([\text{DL+LL+WL}]\) in the Name field.

3. Set the Type option to Normal.

   This combines the loads algebraically.

   The other combination types available are called SRSS (square root of sum of squares) and ABS (Absolute). The SRSS type offers the flexibility of part SRSS and part Algebraic. That is, some load cases are combined using the square root of sum of squares approach, and the result is combined with other cases algebraically, as in:

   \[ A + \sqrt{B^2 + C^2} \]

   where \( A, B, \text{ and } C \) = the individual primary load cases

4. Type 0.75 in the Default \( \alpha_1 \) field.

   This is used as the default factor for normal load combinations.

5. Click [>>].

   Both primary load cases are added to the Load Case Definition list using the default factor.

   This indicates that the results of the two load cases will be multiplied by 0.75 and then summed to produce load combination results. Note that such a load combination method should only be used for linear analysis.
6. Click Add.
   The load combination is added to the **Load Cases Details** section in the **Load & Definition** dialog as load 3.

7. Click Close.

**T.1 Analysis and Design**

The following procedures describe how to add analysis and design commands to the STAAD input file.

**Note:** The analysis design commands are added using either the analytical user interface or the STAAD.Pro Editor. The STAAD.Pro Physical Modeler interface is not used to assign analysis and design commands.

**T.1 Specifying the analysis type**

A linear, static analysis is required for this model. You can also instruct STAAD.Pro to provide a static equilibrium report.

The STAAD input file commands generated are:

```
PERFORM ANALYSIS PRINT STATICS CHECK
```

1. On the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group.

The **Analysis/Print Commands** dialog opens.

**Tip:** If the **Analysis/Print Commands** dialog does not open automatically, then click **Define Commands** in the **Analysis - Whole Structure** dialog.

2. In the **Analysis/Print Commands** dialog, select the **Perform Analysis** tab.
3. Select the Statics Check option.

**Note:** In response to this option, a report consisting of the summary of applied loading and summary of support reactions, for each load case, will be produced in the STAAD output file.

4. Click Add.

**Tip:** The command is added to the list of commands outlined in the Analysis - Whole Structure dialog on the right-hand side of the window. The command has a green checkmark to indicate that it is valid. You can review the outline of the command input structure here.

5. Click Close.

**Tip:** Remember to save your work by either click **Save** on the File ribbon tab, the **Save** tool, or pressing `<CTRL+S>`.

T.1 Specifying post-analysis print commands

Once an analysis has been completed, the program can then provide results of the analysis in the output. In particular, you will request member end forces and support reactions written to the output file.
The STAAD input file commands generated are:

```
PRINT MEMBER FORCES ALL
PRINT SUPPORT REACTION LIST 1 4
```

1. Select all the beam members by one of the following methods:
   - on the Select ribbon tab, select the All tool in the Beams group
   - click-and-drag a window around all members in the View window
   - press <Ctrl+A>

2. On the Analysis and Design ribbon tab, select the Post-Analysis Commands tool in the Analysis Data group.

3. On the Post Analysis Print - Whole Structure dialog, click Define Commands. The Analysis/Print Commands dialog opens.

4. Add the member forces results to the output:
   - a. Select the Member Forces tab.
b. Click **Assign**.

**Tip:** Since you first selected the members, you can assign the command to those members directly from this dialog. Otherwise, you would add the command and then assign it —similar to how you assigned load case items— to those members.

c. Click **Close**.

5. On the **Select** ribbon tab, select the **Supports** tool in the **Nodes** group.

The supported nodes (nodes 1 and 4) are both selected.

6. Repeat steps 2 and 3 to open the **Analysis/Print Commands** dialog for the post-print commands.

7. Add the support reactions to the output:

a. Select the **Support Reactions** tab.
b. Click **Assign**.

c. Click **Close**.

The command is added to the list of commands.
Figure 25: The **Post Analysis Print** dialog after the post-analysis print commands are added and assigned.
**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL +S>`.

### T.1 Short-listing the load cases to be used in steel design

The steel design has to be performed for load cases 1 and 3 only per the specification at the beginning of this tutorial. To instruct the program to use just these cases, and ignore the remaining, you will use the **LOAD LIST** command.

The STAAD input file commands generated are:

```
LOAD LIST 1 3
```

1. On the **Analysis and Design** ribbon tab, select the **Load List** tool in the **Analysis Data** group.

   The **Load List** dialog opens.

   ![Load List Dialog](image)

   *Figure 26:*

2. From the **Load Cases** list box on the left, double-click on **1: DEAD + LIVE** and **3: 75 Percent of [DL+LL+WL]** to add these loads to the **Load List** box on the right.

3. Click **OK**.

### T.1 Specifying steel design parameters

Designing members per a building code as specified for this tutorial require you to provide values for some of the terms where the default values do not match the problem requirements.

**Note:** This tutorial uses the 2010 edition of the American Institute of Steel Construction 360 specification for design (AISC 360-10). Details on the code parameters can be found in [Section 1 of the Design Codes](#) (on page 1366) section.
The STAAD input file commands generated are:

```
PARAMETER
CODE AISC UNIFIED 2010
FYLD 7200 ALL
TRACK 2 MEMB 2 3
UNB 10.0 MEMB 2 3
UNT 10.0 MEMB 2 3
SELECT MEMB 2 3
```

1. On the Analysis and Design ribbon tab, select Steel in the Design Commands gallery.

2. In the Steel Design - Whole Structure dialog, select AISC 360-10 in the Current Code list.

3. In the Steel Design - Whole Structure dialog, click Define Parameters. The Design Parameters dialog opens.
4. Specify the yield strength parameter:
   a. Select the FYLD parameter tab.
b. Type 7200 (kip/ft$^2$) [50 (kip/in$^2$)] in the Yield strength of steel field.

Tip: This equates to a 50 ksi steel, which is the common strength for American wide flange shapes. You can change the input units before and after this command to use inches for convenience, but this tutorial uses a non-standard unit here for brevity.

c. Click Add.
The parameter list is added to the list of commands in the Steel Design - Whole Structure dialog, including the selected design code and the yield strength value.

5. To define the remaining parameters, repeat step 4 except for selecting the parameters and providing the values listed below.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRACK</td>
<td>2 (select the option)</td>
</tr>
<tr>
<td>UNB</td>
<td>10 (ft) [120 (in)]</td>
</tr>
<tr>
<td>UNT</td>
<td>10 (ft) [120 (in)]</td>
</tr>
</tbody>
</table>

6. Click Close in the Design Parameters dialog.

Note: The steel design parameters are all marked with an icon. This indicates that they need to be assigned to steel members.

7. Assign the yield strength to all members:
   a. Select the FYLD 7200 parameter in the command list.
b. Select the **Assign to View** option in the Assignment Method group.

c. Click **Assign**.

A dialog opens confirming you want to assign to the view (all members currently displayed in the view window).

d. Click **Yes**.

8. Assign the output detail (TRACK) parameter to members 2 and 3:

   a. Select the **TRACK 2** parameter in the command list.

   b. Select the **Use Cursor to Assign** option in the Assignment Method group.

   c. Click **Assign**.

      The mouse pointer changes to a ⬅️.

   d. Click on each member in the Frame.

   e. Click on **Assigning** to stop assigning design parameters.

9. Repeat step 8 to assign the other two parameters (UNB and UNT) to members 2 and 3, as well.
Figure 27: The **Steel Design - Whole Structure** dialog after the design parameters have been added and
10. Add a command to instruct select sizes for members 2 and 3:

The select command is an instruction to the program to assign the least-weight cross-section which satisfies all the code requirements for the member.

a. In the **Steel Design - Whole Structure** dialog, click **Commands**.

The **Design Commands** dialog opens.

b. Select the **Select** tab.

c. Click **Add**.

d. Click **Close**.

11. Assign the select command to members 2 and 3:

a. Select the **SELECT** parameter in the command list.

b. Select the **Assign to Edit List** option in the Assignment Method group.

c. Type **2 3** in the list.

Notice this is a space-separated list of member numbers.

d. Click **Assign**.

**Tip:** You may also use either the method of selecting the members first or using the cursor to assign the command to the members.

After the parameters are assigned, click anywhere in the drawing area to deselect the members.

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL+S>`.
T.1 Re-specifying the analysis command

When the analysis and design engine executes the member selection operation you specified in the previous step, a new set of properties will end up being assigned to those members. This changes the stiffness distribution for the entire structure. Since the structure is statically indeterminate, it should be analyzed again to determine the correct the nodal displacements, member forces, etc. to reflect this new stiffness distribution.

The STAAD input file commands generated are:

```
PERFORM ANALYSIS
```

1. Select the Select command in the Analysis - Whole Structure dialog outline of the model commands.
2. On the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group.

The Analysis/Print Commands dialog opens.

**Tip:** If the Analysis/Print Commands dialog does not open automatically, then click Define Commands in the Analysis - Whole Structure dialog.

3. Select the Perform Analysis tab.
4. Select the No Print option.

**Tip:** The statics check report does need to be repeated.

5. Check the After Current option.
   This allows you to specify that where the new command is added. This will place the new Perform Analysis after the command selected in Step 1, instead of in the first valid point within the model file.
6. Click Add.
7. Click Close.

**Tip:** Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL+S>.

T.1 Re-specifying the TRACK parameter

The final calculation we need to do is make sure the current set of member properties pass the code requirements based on the up-to-date member forces.

This will require that we do a code checking operation again. To restrict the output produced to a reasonable level, we specify the TRACK parameter again but use a different value.

The STAAD input file commands generated are:

```
PARAMETER
CODE AISC UNIFIED 2010
TRACK 0 ALL
```

1. On the Analysis and Design ribbon tab, select Steel in the Design Commands gallery.
2. In the Steel Design - Whole Structure dialog, click Define Parameters.
   The Design Parameters dialog opens.
3. Specify the track parameter:
   a. Select the TRACK parameter tab.
   b. Select option 0.
      
      **Note:** The TRACK 0 command instructs the program to only provide the controlling limit state check for each member assigned to this command.
   c. Click Add.
      The parameter list is added to the list of commands in the Steel Design - Whole Structure dialog, including the selected track parameter and value.
   d. Click Close.
4. Select the TRACK 0 parameter entry in the list of commands.
5. Select all the beam members by one of the following methods:
   on the Select ribbon tab, select the All tool in the Beams group

   or

   click-and-drag a window around all members in the View window
   or

   press <Ctrl+A>
6. In the Steel Design - Whole Structure dialog, select the Assign To Selected Beams option.
7. Click Assign.
   A message dialog prompting you to confirm the assignment.
8. Click Yes.

After the parameters are assigned, click anywhere in the drawing area to deselect the members.

**Tip:** Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL+S>.

**T.1 Specifying the CHECK CODE command**

As part of the iterative process of analysis and design, you must perform a code check on the selected members after they have been analyzed. This ensures that the new set of member forces do not exceed the capacity of the selected members.
A code checking operation which uses the up-to-date cross sections of the members and the latest member forces is used to evaluate that the members have sufficient capacity per the code specifications.

The STAAD input file commands generated are:

CHECK  CODE  ALL

1. In the Steel Design - Whole Structure dialog, click Commands. The Design Commands dialog opens.
2. Select the Check Code tab.

![Design Commands dialog](image)

3. Click Add.
4. Click Close.
5. Select the unassigned CHECK CODE entry in the Steel Design - Whole Structure dialog.
6. Select the Assign to View option in the Assignment Method group.
7. Click Assign. A message dialog prompting you to confirm the assignment.
8. Click Yes.

Tip: Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL+S>.
T.1 Viewing the input command file

You can inspect the text data file created during this tutorial.

1. On the Utilities ribbon tab, select the Command File tool in the Edit group.

The STAAD.Pro Editor opens with the contents of the input file.

2. (Optional) You modify the data of the structure in this editor if necessary.

3. Select File > Exit Editor in the STAAD.Pro Editor window to close.

As stated in "Methods of creating the model" (on page 373), you could also have created the same model by typing the relevant STAAD commands into a text file using the STAAD.Pro Editor. If you would like to understand that method, proceed to the next section.

If you want to skip that part, proceed to "Performing Analysis and Design" (on page 432) where you will perform the analysis and design on this model.
T.1 Creating the model using the command file

As an alternative to the procedures described in the preceding tutorial, you can also create the same STAAD input file using the STAAD.Pro Editor.

**Note:** A STAAD input file is a plain text file that uses the `.std` file extension. Therefore any standard text editor such as Notepad can also be used to create the command file. However, the STAAD.Pro Editor offers the advantage of syntax checking as you type the commands. The STAAD command syntax are highlighted by command, keyword, value, etc.

To start a new STAAD input file using the STAAD.Pro Editor, follow the procedure described in “Creating a new structure” (on page 375), except select the **Open STAAD Editor** check box in step 7. The STAAD.Pro Editor window opens with the basic commands for your model entered.

For this tutorial, delete all the command lines displayed in the editor window and type the lines shown below. While not necessary, this will allow you to learn more about the required and optional command lines for an input file.

STAAD commands are **not** case sensitive (i.e., they may be typed in upper or lower case letters). By convention, this and most input files use all caps, though.

For most all commands and keywords, the first three letters of a keyword are all that are needed. The rest of the letters of the word are not required, but are useful to present a user-friendly command language in mostly plain
English for later reference. By convention, the required letters in a command or keyword are underlined here ("PLANE" = "PLA" = "plane" = "pla").

**STAAD PLANE PORTAL FRAME**

Every STAAD input file has to begin with the word **STAAD**. The word **PLANE** signifies that the structure is a plane frame (in the XY plane). The remainder of this line is the title of the problem, which is optional.

**Note:**

If a line is typed with an asterisk in the first column, it signifies that the line is a comment line and should not be executed. For example, one could have put the optional title above on a separate line as follows:

```
* PORTAL FRAME
```

**UNIT FEET KIP**

Specify the force and length units for the commands to follow.

**JOINT COORDINATES**

```
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
```

Joint numbers and their corresponding global X and Y coordinates are provided above. For example, 3 20 15. indicates that node 3 has an X coordinate of 20 ft and a Y coordinate of 15 ft. Note that the reason for not providing the Z coordinate is because the structure is a plane frame. If this were a space frame, the Z coordinate would also be required.

Semicolons (;) are used as line separators. In other words, data which is normally put on multiple lines can be put on one line by separating them with a semicolon.

**MEMBER INCIDENCE**

```
1 1 2 ; 2 2 3 ; 3 3 4
```

The members are defined by the joints to which they are connected.

**MEMBER PROPERTY AMERICAN**

```
1 3 TABLE ST W12X35
2 TABLE ST W14X34
```

Members 1 and 3 are assigned a W12X35 section from the built-in AMERICAN steel table. Member 2 has been assigned a W14X34. The word ST stands for standard single section. Sections 5.20.1 through 5.20.5 of the STAAD Technical Reference help explain the convention for assigning member property names.

**UNIT INCHES**

**DEFINE MATERIAL START**

```
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

The length unit is changed from FEET to INCHES to use familiar units for most of the material definition. For this example, you use a set of built-in values for steel, so the units are only shown for convenience here. In the user
interface, units were not changed until the following commands were generated. See Section 5.26.1 of the STAAD Technical Reference help for more information.

```
MEMBER OFFSET
  2  START  6.0  0.  0.
  2  END   -6.0  0.  0.
```

The beam member is physically connected to the 2 columns at the face of the column, and not at the column centerline. This creates a rigid zone, about half the depth of the columns, at the 2 ends of the beam 2. This rigid zone is taken advantage of using member offsets (It is you choice whether or not you wish to use these). So, the above commands define that member 2 is eccentrically connected or OFFSET at its START joint by 6 inches in the global X direction, 0.0 and 0.0 in Y and Z directions. The same member is offset by negative 6.0 inches at its END joint. See Section 5.25 of the STAAD Technical Reference help for more information.

```
PRINT MEMBER INFORMATION ALL
```

The information that is printed by this command includes start and end joint numbers (incidence), member length, beta angle and member end releases.

```
SUPPORTS
  1  FIXED ; 4  PINNED
```

A fixed support is located at joint 1 and a pinned support (fixed for translations, released for rotations) at joint 4. More information on the support specification is available in Section 5.27 of the STAAD Technical Reference help.

```
UNIT FT
```

The length unit is changed to FEET to facilitate input of loads.

```
LOADING 1 DEAD + LIVE
MEMBER LOAD
  2  UNI  GY -2.5
```

The above commands identify a loading condition. DEAD + LIVE is an optional title to identify this load case. A Uniformly distributed MEMBER LOAD of 2.5 kips/ft is acting on member 2 in the negative global Y direction. Member Load specification is explained in Section 5.32 of the STAAD Technical Reference help.

```
LOADING 2 WIND FROM LEFT
JOINT LOAD
  2  FX  10.
```

The above commands identify a second load case. This load is a JOINT LOAD. A 10 kip force is acting at joint 2 in the global X direction.

```
LOAD COMBINATION 3 75 PERCENT OF (DL+LL+WL)
  1 0.75  2 0.75
```

This command identifies a combination load with an optional title. The second line provides the components of the load combination case - primary load cases and the factors by which they should be individually multiplied.

```
PERFORM ANALYSIS PRINT STATICS CHECK
```

This command instructs the program to proceed with the analysis and produce a report of static equilibrium checks. Section 5.37 of the STAAD Technical Reference help offers information on the various analysis options available.

```
PRINT MEMBER FORCES ALL
PRINT SUPPORT REACTION LIST 1 4
```
The above print commands are self-explanatory. The member forces are in the member local axes while support reactions are in the global axes.

```
LOAD LIST 1 3
PARAMETERS
  CODE AISC UNIFIED 2010
  FYLD 7200 ALL
  TRACK 2.0 MEMB 2 3
  UNB 10.0 MEMB 2 3
  UNT 10.0 MEMB 2 3
SELECT MEMBER 2 3
```

The above sequence of commands is used to initiate the steel design process. The command PARAMETERS is followed by the various steel design parameters. Parameters are specified typically when their values differ from the built-in program defaults. Specifications of the AISC Unified Code LRFD specification are to be followed. A parameter list for the AISC code is available in Section 3 of the Technical Reference help. ALL members have 10 ft unsupported length for the top and bottom flange (UNT and UNB).

UNT and UNB are used to compute the allowable compressive stress in bending. The yield strength of steel is specified as 7,200 ksf (50 ksi) since it is different from the default value of 36 ksi. The TRACK parameter controls the level of description of the output, 2.0 being the most detailed. The LOAD LIST command lists the load cases (1 and 3) to be used in the design. The SELECT MEMBER command asks the program to evaluate the most economical section for members 2 and 3 in the context of the above analysis.

PERFORM ANALYSIS

When the analysis & design engine executes the member selection operation specified in the member selection, a new set of properties will end up being assigned to those members. This has the effect of changing the stiffness distribution for the entire structure. Since the structure is statically indeterminate, it is best practice to re-analyze it to determine the accurate nodal displacements, member forces, etc. which reflect this new stiffness distribution. The above command instructs the program to do another cycle of analysis.

```
PARAMETER
  TRACK 0 ALL
```

The TRACK parameter is re-specified. It controls the level of information produced in the steel design output. This time, the value of 0 is used to provide a pass or fail status for each member.

CHECK CODE ALL

The analysis operation carried out earlier will create a new set of member forces. These forces will very likely be different from those which were used in the member selection operation. Consequently, you should verify that the structure is safely able — from the standpoint of the design code requirements — to carry these new forces. A code checking operation, which uses the up-to-date cross sections of the members, and the latest member forces, will provide a status report on this issue.

FINISH

A STAAD run is terminated using the FINISH command.

Save the input file and close the editor. The model is opened in the STAAD.Pro interface.

This concludes the session on generating the model as a command file using the built-in editor. If you wish to perform the analysis and design, you may proceed to the next section of this manual (on page 432). The post-processing facilities are explained in Post Processing (on page 441).

**Caution:** Remember that without successfully completing the analysis and design, the post-processing facilities will not be accessible.
T.1 Performing Analysis/Design

STAAD.Pro performs Analysis and Design simultaneously.

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL + S>.

1. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.

The **STAAD Analysis and Design** dialog opens and displays messages of the analysis progress.

![STAAD Analysis and Design dialog](image)

2. Select the **View Output File** option once the analysis and design are complete.

The three options are indicative of what will happen after you click **Done**.
View Output File

This option opens the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands specified during the model generation process. It also provides you with important messages of any errors that were encountered, and if so, whether the analysis and design was successfully completed or not. See T.1 Viewing the output file (on page 433) for details on viewing and understanding the contents of the output file.

Go to Post Processing Mode

This option opens the graphical Post-processor mode, which can be used to extensively review and verify the results. This mode allows you to view the results graphically, plot result diagrams, produce reports, etc. See T.1 Post-Processing (on page 441) for details on the Post processing mode.

Stay in Modeling Mode

This option closes the dialog and remains in the Model generation mode of the program, where you initiated the analysis. This is useful if you want to make further changes to the input file.

3. Click Done.
   The STAAD.Pro Output Viewer window opens.

T.1 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly.

For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

**Note:**
If you did not select to open the output file after running the analysis in the previous procedure, you can open it any time through the user interface. On the **Utilities** ribbon tab, select the **Analysis Output** tool in the **View** group.

**Tip:** By default, the output file contains a listing of the entire input also. You may choose not to print the echo of the input commands in the Output file. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands > Set Echo** tool option from the menu bar and select the **Echo Off** option in the **Set Echo** dialog.

It is **strongly recommended** that you review the entire output file to ensure that the results are reasonable and that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the post-processing mode. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.
1. STAAD PLANE PORTAL FRAME
INPUT FILE: Portal.STD
2. UNIT FEET KIP
3. JOINT COORDINATES
4. 1 0 0 0; 2 0 15 0; 3 20 15 0; 4 20 0 0
5. MEMBER INCIDENCES
6. 1 1 2; 2 2 3; 3 3 4
7. UNIT INCHES KIP
8. DEFINE MATERIAL START
9. ISOTROPIC STEEL
10. E 29000
11. POISSON 0.3
12. DENSITY 283E-006
13. ALPHA 6E-006
14. DAMP 0.03
15. TYPE STEEL
16. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
17. END DEFINE MATERIAL
18. MEMBER PROPERTY AMERICAN
19. 1 3 TABLE ST W12X35
20. 2 TABLE ST W14X34
21. CONSTANTS
22. MATERIAL STEEL ALL
23. MEMBER OFFSET
24. 2 START 6 0 0
25. 2 END -6 0 0
26. SUPPORTS
27. 1 FIXED
28. 4 PINNED
29. UNIT FEET KIP
30. LOAD 1 LOADTYPE NONE TITLE DEAD + LIVE
31. MEMBER LOAD
32. 2 UNI GY -2.5
33. LOAD 2 LOADTYPE NONE TITLE WIND FROM LEFT
34. JOINT LOAD
35. 2 FX 10
36. LOAD COMB 3 75 PERCENT OF [DL+LL+WL]
37. 1 0.75 2 0.75
38. PERFORM ANALYSIS PRINT STATICS CHECK
PORTAL FRAME

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2

Using 64-bit analysis engine. 
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       7
TOTAL LOAD COMBINATION CASES =     1 SO FAR.
PORTAL FRAME

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
LOADTYPE NONE TITLE DEAD + LIVE
CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
TOTAL APPLIED LOAD  1
***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING  1 )
SUMMATION FORCE-X =  0.00
SUMMATION FORCE-Y =  -47.50
SUMMATION FORCE-Z =  0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=  0.00  MY=  0.00  MZ=  -475.00

TOTAL REACTION LOAD  1
***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING  1 )
SUMMATION FORCE-X =  -0.00
SUMMATION FORCE-Y =  47.50
SUMMATION FORCE-Z =  0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=  0.00  MY=  0.00  MZ=  475.00

MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING  1)
MAXIMUMS AT NODE
X =  1.82361E-01       2
Y = -1.46578E-02       3
Z =  0.00000E+00       0
RX=  0.00000E+00       0
RY=  0.00000E+00       0
RZ= -4.82523E-03       2

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.  2
LOADTYPE NONE TITLE WIND FROM LEFT
CENTER OF FORCE BASED ON X FORCES ONLY (FEET).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.000000000E+00
Y =  0.150000004E+02
Z =  0.000000000E+00

PORTAL FRAME -- PAGE NO.  4
TOTAL APPLIED LOAD  2
***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING  2 )
SUMMATION FORCE-X =  10.00
SUMMATION FORCE-Y =  0.00
SUMMATION FORCE-Z =  0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=  0.00  MY=  0.00  MZ=  -150.00

TOTAL REACTION LOAD  2
***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING  2 )
SUMMATION FORCE-X =  -10.00
SUMMATION FORCE-Y =  0.00
SUMMATION FORCE-Z =  0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=  0.00  MY=  0.00  MZ=  150.00

MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING  2)
MAXIMUMS AT NODE
X =  7.27292E-01       2
Y =  2.47270E-03       2
Z =  0.00000E+00       0
RX=  0.00000E+00       0
RY=  0.00000E+00       0
RZ= -5.48837E-03       4

************** END OF DATA FROM INTERNAL STORAGE **************
39. PRINT MEMBER INFORMATION ALL
MEMBER INFORMAT ALL
### Member Information

<table>
<thead>
<tr>
<th>Member</th>
<th>Start Joint</th>
<th>End Joint</th>
<th>Length (Feet)</th>
<th>Beta (Deg)</th>
<th>Releases</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>2</td>
<td>15.000</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>3</td>
<td>19.000</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>4</td>
<td>15.000</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

************* END OF DATA FROM INTERNAL STORAGE *************

### Member Forces All

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>Joint</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>23.18</td>
<td>-3.99</td>
<td>0.00</td>
<td>0.00</td>
<td>-11.48</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>-23.18</td>
<td>3.99</td>
<td>0.00</td>
<td>0.00</td>
<td>-48.40</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>14.30</td>
<td>2.77</td>
<td>0.00</td>
<td>0.00</td>
<td>42.34</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>-14.30</td>
<td>-2.77</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.81</td>
<td></td>
</tr>
</tbody>
</table>

************* END OF LATEST ANALYSIS RESULT *************

### Support Reaction List 1

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>3.99</td>
<td>23.18</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-11.48</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>-7.68</td>
<td>-4.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>67.93</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>-2.32</td>
<td>4.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-32.69</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>4.73</td>
<td>14.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>60.31</td>
</tr>
</tbody>
</table>

************* END OF LATEST ANALYSIS RESULT *************

### Load List 1

### Parameter 1

### Code AISC Unified 2010

### Fyld 7200 All

### Track 2 Mem 2 3

### Unb 10 Mem 2 3

### Unt 10 Mem 2 3

### Select Mem 2 3

### Steel Design
### STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD) v1.4a

---

**MEMBER** | **TABLE** | **RESULT/LOADING** | **CRITICAL COND/LOCATION** | **RATIO/LOCATION**
--- | --- | --- | --- | ---
2 ST | W10X22 | PASS | Eq. H1-1b | 0.967 | 1

### SLENDERNESS

- Actual Slenderness Ratio: 172.030 L/C: 3
- Allowable Slenderness Ratio: 200.000 LOC: 0.00

### STRENGTH CHECKS

- Critical L/C: 1 Ratio: 0.967 (PASS) Condition: Eq. H1-1b

### DESIGN FORCES

- Fx: 3.992E+00(C) Fy: -5.738E-01 Fz: 0.000E+00
- Mx: 0.000E+00 My: 0.000E+00 Mz: -7.055E+01

### SECTION PROPERTIES (UNIT: INCH)

- Azz: 4.140E+00 Ayy: 2.448E+00 Cw: 2.760E+02
- Szz: 2.314E+01 Syy: 3.965E+00
- Izz: 1.180E+02 Iyy: 1.140E+01 Ix: 2.390E-01

### MATERIAL PROPERTIES

- Fyld: 7199.999 Fu: 8639.999

### Check for Axial Tension

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>2.92E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>1</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>2.92E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>1</td>
</tr>
</tbody>
</table>

### Check for Axial Compression

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>4.73E+00</td>
<td>2.37E+02</td>
<td>0.020</td>
<td>Eq. E3-1</td>
<td>3</td>
</tr>
<tr>
<td>Min Buck</td>
<td>4.73E+00</td>
<td>4.95E+01</td>
<td>0.095</td>
<td>Eq. E3-1</td>
<td>3</td>
</tr>
<tr>
<td>Flexural</td>
<td>Tor Buck</td>
<td>4.73E+00</td>
<td>1.53E+02</td>
<td>0.031</td>
<td>Eq. E4-1</td>
</tr>
</tbody>
</table>

---

**PORTAL FRAME - PAGE NO. 9**

---

**Tutorials**

T.1 – Steel Portal Frame

---

**STAAD.Pro 437 User Manual**
## Intermediate Results

<table>
<thead>
<tr>
<th>Eff Area</th>
<th>KL/r</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>4.51E-02</td>
<td>53.47</td>
<td>5.84E+03</td>
<td>1.44E+04</td>
</tr>
<tr>
<td>Min Buck</td>
<td>4.51E-02</td>
<td>172.03</td>
<td>1.22E+03</td>
<td>1.39E+03</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flexural Ag</th>
<th>Fcr</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tor Buck</td>
<td>4.51E-02</td>
<td>3.77E+03</td>
</tr>
</tbody>
</table>

---

### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>1.12E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>1</td>
</tr>
<tr>
<td>Local-Y</td>
<td>-2.43E+01</td>
<td>7.34E+01</td>
<td>0.331</td>
<td>Eq. G2-1</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

Major | Minor |
---|---|
| 7.05E+01 | 9.75E+01 |
| 0.00E+00 | 2.29E+01 |

<table>
<thead>
<tr>
<th>Mm</th>
<th>My</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.08E+02</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>2.54E+01</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

---

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>Major</th>
<th>Minor</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.05E+01</td>
<td>7.62E+01</td>
</tr>
<tr>
<td>0.00E+00</td>
<td>2.29E+01</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mn</th>
<th>Me</th>
<th>Cb</th>
<th>Lp</th>
<th>Lr</th>
<th>Lb</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.46E+01</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>4.68</td>
<td>13.84</td>
<td>10.00</td>
</tr>
</tbody>
</table>

---

### CHECK FOR FLEXURE TENS/COMP INTERACTION

Flexure Comp | Flexure Tens |
---|---|
| 7.62E+01 | 7.05E+01 |
| 2.29E+01 | 0.00E+00 |

<table>
<thead>
<tr>
<th>Mcx</th>
<th>Mrx</th>
<th>Pc</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.62E+01</td>
<td>7.05E+01</td>
<td>4.95E+01</td>
</tr>
<tr>
<td>2.29E+01</td>
<td>0.00E+00</td>
<td>3.99E+00</td>
</tr>
</tbody>
</table>

---

### CHECK FOR IN-PLANE/OUT-OF-PLANE FLEXURE COMP INTERACTION

In-Plane Flx Comp | Out Plane Flx Comp |
---|---|
| 6.83E+01 | 7.62E+01 |
| 8.38E+01 | 7.62E+01 |

<table>
<thead>
<tr>
<th>Mr</th>
<th>Mc</th>
<th>Pr</th>
<th>Pc</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.83E+01</td>
<td>7.62E+01</td>
<td>3.99E+00</td>
<td>2.37E+02</td>
</tr>
<tr>
<td>6.83E+01</td>
<td>7.62E+01</td>
<td>3.99E+00</td>
<td>4.95E+01</td>
</tr>
</tbody>
</table>
### 3 ST W10X26 (AISC SECTIONS)

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 ST</td>
<td>W10X26</td>
<td>PASS</td>
<td>Eq. H1-2</td>
<td>0.863</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>21.32 C</td>
<td>0.00</td>
<td>70.97</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### slenderess
- **Actual Slenderness Ratio**: 132.238 L/C: 3
- **Allowable Slenderness Ratio**: 200.000 LOC: 0.00

#### Strength Checks
- **Critical L/C**: 3 Ratio: 0.863(PASS)
  
  Condition: Eq. H1-2

#### Design Forces
- \( F_x = 2.132 \times 10^1 \text{ (C) } \)
- \( F_y = 4.731 \times 10^0 \)
- \( F_z = 0.000 \times 10^0 \)
- \( M_x = 0.000 \times 10^0 \)
- \( M_y = 0.000 \times 10^0 \)
- \( M_z = 7.097 \times 10^1 \)

#### Section Properties (Unit: Inch)
- \( A_{zz} = 5.078 \times 10^0 \)
- \( A_{yy} = 2.678 \times 10^0 \)
- \( C_w = 3.427 \times 10^2 \)
- \( S_{zz} = 2.796 \times 10^1 \)
- \( S_{yy} = 4.887 \times 10^0 \)
- \( I_{zz} = 1.440 \times 10^2 \)
- \( I_{yy} = 1.410 \times 10^1 \)
- \( I_x = 4.020 \times 10^-1 \)

#### Material Properties
- \( F_y = 7199.999 \)
- \( F_u = 8639.999 \)

#### Actual Member Length: 15.000

#### Design Parameters
- \( K_z = 1.00 \)
- \( K_y = 1.00 \)
- \( NSF = 1.00 \)
- \( SLF = 1.00 \)
- \( CSP = 12.00 \)

#### Check for Axial Tension

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>3.42E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>1</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>3.42E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>1</td>
</tr>
</tbody>
</table>

#### Check for Axial Compression

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>2.43E+01</td>
<td>3.02E+02</td>
<td>0.081</td>
<td>Eq. E3-1</td>
<td>1</td>
</tr>
<tr>
<td>Min Buck</td>
<td>2.43E+01</td>
<td>9.83E+01</td>
<td>0.247</td>
<td>Eq. E3-1</td>
<td>1</td>
</tr>
<tr>
<td>Flexural</td>
<td>2.43E+01</td>
<td>2.20E+02</td>
<td>0.110</td>
<td>Eq. E4-1</td>
<td>1</td>
</tr>
<tr>
<td>Results</td>
<td>Eff Area</td>
<td>KL/r</td>
<td>Fcr</td>
<td>Fe</td>
<td>Pn</td>
</tr>
<tr>
<td>--------------</td>
<td>----------</td>
<td>----------</td>
<td>-------</td>
<td>------------</td>
<td>------------</td>
</tr>
<tr>
<td>Maj Buck</td>
<td>5.28E-02</td>
<td>41.38</td>
<td>6.35E+03</td>
<td>2.41E+04</td>
<td>3.36E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>5.28E-02</td>
<td>132.24</td>
<td>2.07E+03</td>
<td>2.36E+03</td>
<td>1.09E+02</td>
</tr>
<tr>
<td>Flexural</td>
<td>Ag</td>
<td>Fcr</td>
<td>Pn</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tor Buck</td>
<td>5.28E-02</td>
<td>4.63E+03</td>
<td>2.45E+02</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**CHECK FOR SHEAR**

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>1.37E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>1</td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.73E+00</td>
<td>8.03E+01</td>
<td>0.059</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
</tbody>
</table>

**CHECK FOR BENDING-YIELDING**

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-7.10E+01</td>
<td>1.17E+02</td>
<td>0.605</td>
<td>Eq. F2-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>2.81E+01</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING**

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-7.10E+01</td>
<td>9.47E+01</td>
<td>0.749</td>
<td>Eq. F2-2</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
</tr>
<tr>
<td>Major</td>
<td>1.05E+02</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>4.81</td>
<td>14.89</td>
</tr>
</tbody>
</table>

**CHECK FOR FLEXURE TENS/COMP INTERACTION**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.341</td>
<td>Eq. H1-1a</td>
<td>1</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.749</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
</tbody>
</table>

**CHECK FOR IN-PLANE/OUT-OF-PLANE FLEXURE COMP INTERACTION**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>In-Plane Flx Comp</td>
<td>0.784</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
<tr>
<td>Out Plane Flx Comp</td>
<td>0.863</td>
<td>Eq. H1-2</td>
<td>3</td>
</tr>
</tbody>
</table>

50. PERFORM ANALYSIS
** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
51. PARAMETER 2
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/LOCATION</th>
<th>CRITICAL COND/LOADING/</th>
<th>RATIO/</th>
<th>FX</th>
<th>MY</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST</td>
<td>W12X35</td>
<td>PASS Eq. H1-1b</td>
<td>0.531</td>
<td>1</td>
<td>23.77 C</td>
<td>0.00</td>
<td>65.68</td>
</tr>
<tr>
<td>2 ST</td>
<td>W10X22</td>
<td>PASS Eq. H1-1b</td>
<td>0.787</td>
<td>1</td>
<td>4.35 C</td>
<td>0.00</td>
<td>-59.21</td>
</tr>
<tr>
<td>3 ST</td>
<td>W10X26</td>
<td>PASS Eq. H1-2</td>
<td>0.808</td>
<td>1</td>
<td>23.73 C</td>
<td>0.00</td>
<td>65.28</td>
</tr>
</tbody>
</table>

55. FINISH

*********** END OF THE STAAD.Pro RUN ***********

PORTAL FRAME                                             -- PAGE NO.   15

*** For technical assistance on STAAD.Pro, please visit ***
* Details about additional assistance from
* Bentley and Partners can be found at program menu
* Help->Technical Support
* Copyright (c) 1997-2017 Bentley Systems, Inc.
* http://www.bentley.com

T.1 Post-Processing

STAAD.Pro offers extensive result verification and visualization facilities. These facilities are accessed from the Post Processing Mode. The Post Processing mode is used to verify the analysis and design results and generate reports.

T.1 Opening the postprocessing workflow

You can open the Postprocessing workflow anytime there are current analysis results for your model.

If you selected to open the Postprocessing workflow after T.1 Performing Analysis/Design (on page 432), then you can skip to step 2.

1. On the Workflows panel, select Postprocessing.
The Results Setup dialog opens.

2. (Optional) Select the load cases for which to display the results.

   **Tip:** All load cases are selected by default.

   For this tutorial, we will use all load cases.

3. Select the Result View Options tab and then check the Enable Automatic Scaling option.

4. Click OK.

   **Note:**
   Notice that in the Postprocessing workflow, the Postprocessing page control bar is opens above the view window. The ribbon bar updates to include the Results ribbon tab.

T.1 Annotating the displacements

Annotation is the process of displaying the displacement values on the screen.

1. Select the Displacements page in the Postprocessing page control bar.

   The diagram displayed is the node deflection diagram for load case 1 (DEAD + LIVE).

2. On the Results ribbon tab, select the Annotate tool in the Configuration group.
The Annotation dialog opens.

3. From the Ranges tab, select All.

   **Tip:** If you want to annotate deflection for just a few nodes, specify the node numbers in the node list.

4. Select the Node tab and then check the Resultant option.

   The resultant is the square root of sum of squares of values of X, Y and Z displacements.
5. Click the **Annotate**.
   The values appear on the structure.

6. Click **Close**.

---

**Figure 31:**

**Figure 32:** The deflected shape diagram for load case 1
T.1 Displaying force and moment diagrams

1. Select the Beam Results page in the Postprocessing page control bar.

   The bending moment MZ is plotted by default. Note that the Mz tool in the View Results group on the Results ribbon tab is depressed (active).

2. (Optional) Change the force or moment diagram drawn on the structure in the Diagrams dialog:
   a. Right-click in the view window and then select Structure Diagrams from the pop-up menu. The Diagrams dialog opens.
   b. Select the Loads and Results tab.
   c. Select a different Load Case from the drop-down list.
   d. Select one or more options in the Beam Forces group.
      For example, select the Shear yy to display the shear force diagrams in the local y direction on the members.
   e. (Optional) Click on the color block adjacent to any of the force or moment options to change the color for that diagram.
   f. (Optional) The Beam Forces Diagram options control how the diagrams are filled.

   Tip: Alternatively, you can select which diagrams are displayed and for which load case using the tools and load cases selection in the View Results group on the Result ribbon tab.

3. On the Results ribbon tab, select the Annotate tool in the Configuration group.
4. From the Ranges tab, select All.
   
   **Tip:** If you wish to annotate the force/moment for just a few members, specify the beam numbers in the beam list.

5. Select the locations where you want to display results:
   
   a. Select the Beam Results tab
   
   b. Check the Ends and Mid point options in the Bending group.
6. Click **Annotate**.

The diagram is updated with annotated values displayed at the selected locations.
7. Click Close.

T.1 Displaying the dimensions of the members

1. On the Utilities ribbon tab, select the Beam Tools > Dimension Beams tool in the Geometry Tools group.

The Display/Remove Dimension dialog opens.

2. Select the Dimension to View option.
3. Click Display.
4. Click Close.
   The dimensions of the members will appear alongside the members.

T.1 Update physical model with design results

If you generated the model geometry using a physical model, then you need to update the physical model with design results.

Since you performed a member selection as part of the member design in the analytical model, the sections need to be updated in the physical model to keep it synchronized. STAAD.Pro Physical Modeler will detect that design results are present and prompt you to do this.

1. Select Physical Modeling workflow in the Workflows panel.

   The STAAD.Pro Physical Modeler window opens and the Member Design Selection dialog opens.

2. Click the Profile selection drop-down.
   The member design results table opens.

   **Note:** Here you can review the different analytical member design results for each physical member. Recall that a physical member may be decomposed into multiple analytical members and each of those may have different design results in STAAD.Pro (unless the FIXED GROUP command is used).
3. Click **Accept**.
The table closes.

4. Click **Commit**.
The dialog closes and the physical model is updated with the new sections from the STAAD.Pro steel design results.

## T.2 - RC Framed Structure

This tutorial provides step-by-step instructions for creating the model of a reinforced concrete framed structure using STAAD.Pro.

### T.2 Methods of creating the model

As explained in "[T.1 Methods of creating the model](on page 373)", there are three methods of creating the structure data:

1. **Using the STAAD.Pro Physical Modeler interface** (on page 452),
2. **using the traditional STAAD.Pro analytical modeling interface** (on page 458),
3. and **using the command file** (on page 495).

All three methods are explained in this tutorial.

### T.2 Description of the tutorial problem

The structure for this project is a 2 bay, 2 story reinforced concrete frame. The figure below shows the structure. The purpose of this tutorial is to create the model, assign all required input, and perform the analysis and concrete design.
**Figure 34:**

**Basic Data for the Structure**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member properties</td>
<td>Beams 2 &amp; 5: Rectangular, 275 mm width × 350 mm depth</td>
</tr>
<tr>
<td></td>
<td>Columns 1 &amp; 4: Rectangular, 275 mm width × 300 mm depth</td>
</tr>
<tr>
<td></td>
<td>Column 3: Circular, 350 mm diameter</td>
</tr>
<tr>
<td>Member Orientation</td>
<td>All members except column 4: Default</td>
</tr>
<tr>
<td></td>
<td>Column 4: Rotated by 90 degrees with respect to default condition</td>
</tr>
<tr>
<td>Material Constants</td>
<td>Modulus of Elasticity: 22 kN/mm^2</td>
</tr>
<tr>
<td></td>
<td>Density: 17 kn/m^3 (lightweight concrete)</td>
</tr>
<tr>
<td></td>
<td>Poisson's Ratio: 0.17</td>
</tr>
<tr>
<td>Supports</td>
<td>Base of all columns: Fixed</td>
</tr>
</tbody>
</table>
### T.2 Creating a new structure

On the Start page **New** tab, you will provide some initial data necessary for building the model.

1. On the Start page, select **New**.
   
   The **New** page opens to the **Model Info** tab.

2. **Type as** **RCFrame** in the **File Name** field.

3. **Specify a Location** where the STAAD input file will be located on your computer or network.

   You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.

---

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Data</th>
</tr>
</thead>
</table>
| **Loads**          | Load case 1: Dead Load  
|                    | Selfweight of the structure.  
|                    | Beams 2 & 5: 400 kg/m in global Y downward  
|                    | Load case 2: Live Load  
|                    | Beams 2 & 5: 600 kg/m in global Y downward  
|                    | Load case 3: Wind Load  
|                    | Beam 1: 300 kg/m along positive global X  
|                    | Beam 4: 500 kg/m along positive global X  
|                    | Load Case 4: DEAD + LIVE  
|                    | L1 X 1.2 + L2 X 1.5 (Use REPEAT LOAD, not Load Combination)  
|                    | Load Case 5: DEAD + WIND  
|                    | L1 X 1.1 + L2 X 1.3 (Use REPEAT LOAD, not Load Combination)  |
| **Analysis Type**  | **PDELTA**                                                                                                                      |
| **Concrete Design**| Consider load cases 4 and 5 only.  
|                    | Parameters: Ultimate Strength of Steel: 415 N/mm²  
|                    | Concrete Strength: 25 N/mm²  
|                    | Clear cover for top: 25 mm  
|                    | Clear cover for bottom: 30 mm  
|                    | Clear cover for side: 25 mm  
|                    | Design beams 2 and 5  
|                    | Design columns 1, 3 and 4  |
4. Select **Physical** for the **Type** of model.
   This option selects the modeling method you want to use:

   **Analytical**  For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.

   **Physical**  For creating a model using the STAAD.Pro Physical Modeler interface.

   **Building**  For creating a building model structure using the Building workflow.

5. Select **Metric** as the system of **Units**.

   **Tip:** The units can be changed later if necessary, at any stage of the model creation.

6. (Optional) Select the Job Info tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.

7. Click **Create**.

   The STAAD.Pro modeling environment opens and your model file is then opened in the STAAD.Pro Physical Modeler.

   Alternatively, if you want to use either the analytical modeling workflow or the command input file editor to create the model, choose **Analytical**, click **Create**, and then proceed to either "T.1 Creating the model using the analytical user interface" or "T.2 Creating the model using the command file" respectively.

T.2 Creating the Model using the Physical Modeler

You are now ready to start building the model geometry. The steps for doing this are described in the following sections.

   **Tip:** Refer to the STAAD.Pro Physical Modeler Application Window Layout for reference on the application window.

T.2 Generate the model geometry

1. On the **Model** ribbon tab, select the **Grid** tool in the **Create** group.

   The **Create Grid** dialog opens.

2. Supply the grid details:
   a. Type Grid 1 in the **Name** field.
   b. Leave the **Plane** selection as XY and the **Creation Method** as By Spacing.
   c. Type 6 in the **Number of spaces in X** and 1 m in the **Grid spacing X** fields.
   d. Type 7 in the **Number of spaces in Y** and 0.5 m in the **Grid spacing Y** fields.
3. Use the **Member** tool to create the first portal frame:
   a. On the **Model** ribbon tab, select the **Member** tool in the **Create** group.

   ![Member tool]

   b. Click on the following points on Grid 1, in sequence.
      Double-click the final point to stop creating connected members.
      
      \[
      (0,0,0) \\
      (0,3.5,0) \\
      (6,3.5,0) \\
      (6,0,0) \\
      \]

   c. Select the **Member** tool again to deactivate the tool.

   **Note:** Refer to [T.1 Generating the model geometry](on page 377) in Tutorial 1 for detailed steps on how to draw a portal frame.

4. On the **View** ribbon tab, select the **Grids** tool in the **Reference** group.

   ![Grid tool]

   The tool is no longer highlighted and all grids are hidden in the display.

5. Switch on the node and beam labels:
   a. On the **View** ribbon tab, select the drop-down list below the **Numbering** tool in the **Model** group.

   ![Numbering tool]

   b. Select **Node** and **Member** on this list.
      They are both marked with a check.

   c. Select the **Numbering** tool.

6. On the **Model** ribbon tab, select the **Circular Copy** tool in the **Edit** group.

   ![Circular Copy tool]

   The **Copy Model Circular** dialog opens.

7. Specify the circular copy details:
   a. Leave the **Method** selection as **Global**.
   b. Select **Y** as the **Axis of rotation**.
   c. Type \(6 \text{ m}\) in the Pivot point **X** field.
   d. Type \(90 \text{ deg}\) in the **Arc angle** field.
   e. Type **1** in the **Copies** field.
   f. Click **OK**.
The dialog closes and the selection elements of the portal frame are copied at 90° from the right end. 

8. In the lower, right corner of the view window, click the **Switch to isometric view** tool to display the frame in an isometric view.

**T.2 Assign user-defined concrete material**

If you don’t still have all the members selected, press `<Ctrl+A>`.

1. On the **Member Tools** ribbon tab, select the **Material** tool in the **Assign properties** group.

   The **Assign material** dialog opens.

2. Select the concrete specification:
   a. Select **Custom** from the **Type** drop-down list.
      
      Leave the **Source** as **Catalog** and the **Country** as **United States**.
   b. Select **CONCRETE_ACI318** from the **Material type** drop-down list.

3. Define the custom lightweight concrete material:
   a. Type 0.17 in the **Mu** field (Poisson’s ratio).
   b. Type 0.1E-6 in the **Alpha** field (coefficient of thermal expansion).
   c. Type 0.05 in the **CDAMP** field (composite damping ratio)
   d. Type 25 N/mm2 in the **Fck** field (concrete ultimate strength).
   
      **Tip:** You can type any appropriate unit in any unit system into the field and it will automatically be converted to the current default units.

   e. Type 19080 N/m3 in the **Gamma** field (material density).
   f. Type **CONCRETE-LTW** in the **Name** field.
   g. Click **OK**.

   The **CONCRETE-LTW** material is assigned to all the members. This material is also added to the model catalog for later use if necessary.

On the **Model** ribbon tab, select the **Catalog** tool in the **Catalog** group. The model catalog window opens.

Select the **Material** tool so it is highlighted in the display control toolbar found in the lower-left corner. The **CONCRETE-LTW** material is shown as a tile in the Materials section. Select this tile to review the material values. You can use this interface to make changes to the custom materials and sections in your model.

Once you are done reviewing, click the back arrow in the top-left corner of the window.

**T.2 Assign the member properties**

1. While holding `<Ctrl>`, click on both the end columns (M1 and M4).
2. On the **Member** ribbon tab, select the **Section** tool in the **Assign properties** group.
The Assign Section dialog opens.

3. Define the rectangular column section:
   Leave the Source selection as Catalog.
   a. Select Prismatic in the Type drop-down list.
      Leave the Source selection as Catalog.
   b. Select Solid Rectangle in the Shape drop-down list.
   c. Select RECT in the Template drop-down list.
   d. Type 300 mm in the YD field and 275 mm in the ZD field.
   e. Type RECT-COL in the Name field.
   f. Click OK.

4. Repeat steps 1-3 except to define a 350mm (depth, YD) by 275 mm (width, ZD) rectangular prismatic section named RECT-BEAM to the beam members (M2 and M5).

5. Repeat steps 1-3 again except to define a 350mm diameter (YD) solid circular prismatic section named CIRC-COL to the middle column (M3).

6. Rotate the right column by 90 deg (Beta angle):
   a. Select the right column (M4).
   b. On the Member Tools ribbon tab, select the Rotate tool in the Edit group and then select Rotate 90° from the list.

The column is rotated about its local 1 axis by 90°.

c. On the View ribbon tab, select the Local Axes tool in the Model group.

You can confirm that the local z axis of M4 is now rotate 90° away from the global Z axis.

T.2 Assign supports

1. Orient the view to front view:
   a. (Optional) If the rotation controls are hidden, click the rotation widget in the lower-right corner of the view window.
   b. Select the Switch to front view tool.
   c. (Optional) Click the rotation widget again to hide the controls.
Tip: Alternatively, you can press and hold the right mouse button and drag the mouse pointer to quickly rotate the view so it is easy to select the column bases.

2. Click and drag a box from the right side of the to the left to enclose the column base nodes (N1, N4, and N5).

3. On the Node ribbon tab, select the Fixed tool in the Supports group.

A fixed support is assigned to the selected nodes.

T.2 Assign loads to load cases

1. On the Spreadsheet ribbon tab, select the Load Cases tool in the References group.

The Load Cases spreadsheet opens.

2. For Load 1:
   a. Type Dead Load in the description field of Load case 1.
   b. Select Dead as the Type.
   c. Type -1 in the Self-weight multiplier in field Y.

   Note: This value is negative.

   d. Press <Enter>.
3. Select the beam members (M2 and M5).
4. Add the uniform distributed dead load:
   a. On the Member ribbon tab, select the Distributed tool in the Loads group.

   The Add Member Distributed Load dialog opens. The Load group is already selected to use Load case 1.
   b. Select the Load type as Uniform.
   c. Select Global Y as the Direction.
   d. Type \(-400 \text{ kg/m}\) in the Magnitude field.

   Note: This value is negative and uses different units than the default, so be sure to type out the units as indicated above. It will be automatically converted into the default units of kN/m.
   e. Click OK.

5. Create the live load group:
   a. On the Model ribbon tab, select the Load Case tool in the Load group.

   The Add Load Case dialog opens.
   b. Type Live Load in the Description field.
   Leave the Name as Load Case 2.
   c. Select Live as the Type.
   Leave the self weight modifiers all as zero.
   d. Click OK.

   The live load case is added and selected as the active load case as indicated in the application status bar.

6. Repeat steps 3 and 4 except to add a uniform distributed load of \(-600 \text{ kg/m}\) (in the Global Y direction) to the beam members (M2 and M5).

7. Repeat steps 5 and 6 except to add a Load case 3 with a description of Wind Load and type of Wind.
   This load group will have the following to load items:
   • a uniform force of \(300 \text{ kg/m}\) in the Global X direction and is applied to the left column (M1).
   • a uniform force of \(500 \text{ kg/m}\) in the Global X direction and is applied to the right column (M4).

   Note: As this model uses a second-order, P-Delta analysis, load combinations are not recommended. Instead, repeat loads will be used to combine the load cases so that the combinations are processed by the analysis engine directly. These Repeat loads will be added in the Analytical Modeling workflow.

   Refer to TR.37.2 P-Delta Analysis Options (on page 2797) for further details on load types appropriate for P-Delta analysis.

T.2 Generate the Analysis Model

You have now completed the modeling portion of this tutorial. You will now send the model data back to STAAD.Pro where you will create load combinations, add output commands, select analysis criteria, and specify design parameters.
For this tutorial, you will use the default options for analytical model generation. However, for advanced models you can review and change those options in the Configuration dialog.

1. Select Save to save the physical model.
2. Either:
   - On the Model ribbon tab, select the Return to Analytical Modeling tool in the STAAD.Pro group
   - On the File ribbon tab, select Create analysis model in the backstage tabs

The Return to Analytical Modeling dialog opens and displays the progress of the analytical model being generated.

3. Click OK.
   - If you have not previously associated this model with a CONNECT Project, you are asked to do so. This is will not be necessary for this tutorial.
4. Click Cancel in the Assign Project dialog.

The STAAD.Pro Physical Modeler window closes and the model is loaded into the STAAD.Pro analytical modeling interface.

### T.2 Creating the model using the analytical user interface

The following procedures describe how to use the traditional STAAD.Pro analytical user interface to build the model. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

**Tip:** Refer to GS. Application Window Layout (on page 50) for reference on the application window.
T.2 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc.

The STAAD input file commands generated are:

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 3.5 0.0
3 6.0 3.5 0.0 ; 4 6.0 0.0 0.0
5 6.0 0.0 6.0 ; 6 6.0 3.5 6.0

MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4 ; 4 5 6 ; 5 3 6
```

When starting in the Analytical Modeling workflow, a grid is initially displayed in the main view window. This grid is controlled by the **Snap Node/Beam** dialog. The directions of the global axes (X, Y, Z) are represented in the icon in the lower left hand corner of the drawing area.

![Figure 35: The STAAD.Pro window with the Snap Node/Beam dialog open](image)

**Note:** The **Snap Node/Beam** dialog opens on the right side of the screen. You can reopen this dialog by selecting the **Snap Grid Beam** tool in the **Structure** group on the **Geometry** ribbon tab.

1. On the **Snap Node/Beam** dialog, click **Create**.
   A dialog opens which will enable us to set up a grid.
   Within this dialog, there is a drop-down list from which you can select Linear, Radial, or Irregular form of grid lines.
Linear used to place the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board.

Radial used to place construction lines to appear in a spider-web style, which makes it easy to create circular type models where members are modeled as piece-wise linear straight line segments.

Irregular used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane.

2. Select Linear, which is the Default Grid.

In this structure, the portion consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane. Leave X-Y as the Plane of the grid. The size of the model that can be drawn at any time is controlled by the number of Construction Lines to the left and right of the origin of axes, and the Spacing between adjacent construction lines.

3. Type a Name of Grid 1.

4. Type 12 as the number of lines to the Right of the origin along X and 7 above the origin along Y.

5. Type a Spacing of 0.5 m between lines along both X and Y (see figure below) we can draw a frame 6m X 3.5m, adequate for our segment.

6. Click OK.

7. In the Snap Node/Beam grids list, check the new Grid 1 option.
You can create any number of grids. By providing a name, each new grid can be identified for future reference.

**Tip:** Please note that these settings are only used to generated construction lines. These construction lines enable you to easily draw the structure but do not restrict our overall model to those limits.

**Tip:** To change the settings of this grid, select the name in the Snap Node/Beam dialog and then click **Edit**.

The check by **Default Grid** is automatically cleared. STAAD.Pro will only display a single grid at a time.

8. In the View window, click at the origin (0, 0) to create the first node.
A line is "rubber-banded" between this node and the mouse pointer, which previews the member to placed with the next mouse click.

The Snap Node/Beam feature is active by default.

9. Click on the following points to create nodes and automatically join successive nodes by beam members.

(0, 3.5)
(6, 3.5)
(6, 0)

10. Click Close in the Snap Node/Beam dialog.
The grid is hidden.
It is very important that we save our work often, to avoid loss of data and protect our investment of time and effort against power interruptions, system problems, or other unforeseen events.

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing <CTRL +S>.

**T.2 Copying Model Objects by Circular Repeat**

By examining the structure diagram for this tutorial exercise, you may have observed that members 4 and 5 can be easily generated if we could first create a copy of members 1 and 2 and then rotate those copied units about a vertical line passing through the point (6, 0, 0, that is, node 4) by 90 degrees. Fortunately, such a facility does exist which can be executed in a single step. It is called **Circular Repeat** and is available under the **Geometry** menu.

1. Right click anywhere in the view window and select **Labels** from the pop-up menu.

   The **Diagrams** dialog opens to the **Labels** tab.

2. Set the **Node Numbers** and **Beam Numbers** on and then click **OK**.
Tip: Alternatively, you can press <Shift+N> and then <Shift+B> to quickly toggle the same labels. The letters in parenthesis on the Labels tab indicate these <Shift> key shortcuts.

3. (Optional) You can change the font of the labels by selecting File > Settings > Display Options and then changing the corresponding label Font size in the Options dialog.

4. Select the first column and beam for copying:
   a. On the Select ribbon tab, select the Beam Cursor tool in the Cursors group.
b. Holding the <Ctrl> key, click on members 1 and 2.

5. On the Geometry ribbon tab, select the Circular Repeat tool in the Structure group.

The 3D Circular dialog opens.

6. Specify the circular repeat details:
   a. Select Y as the Axis of Rotation.
   b. Type 4 in the Node field.

   This is the node number through which the rotation axis will pass.

   **Tip:** You can click the Node Selection tool here and then click on the node you want the rotation axis to pass through in the View window.

c. Type 6 in the point X Coordinate field and type 0 in the point Z Coordinate field.

d. Type 90 in the Total Angle field.

e. Set the No of Steps value to 1.

f. Clear the Link Steps, Geometry Only, and Use this as the Reference Point for the Beta angle generation options.

g. Click OK.

7. Click anywhere away from the members in the View window to deselect the members.
Figure 39: The frame members drawn

Tip: Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL + S>.

T.2 Changing the input units of length

For specifying member properties for the structure, it is more convenient to use length units are millimeter instead of meter. This requires changing the current length units of input.

The STAAD input file commands generated are:

UNIT MMS KN

1. On the Geometry ribbon tab, select the Input Units tool in the Structure group.

The Set Input Units pop-up opens.
2. Select Metric as the Input Units.
3. Set the Length Units to Milimeter. Leave the Force Units set to KiloNewton.
4. Click Apply.

T.2 User-defined concrete material

The STAAD input file commands generated are:

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE-LTW
E 22
POISSON 0.17
DENSITY 1.9e-008
TYPE CONCRETE
STRENGTH RY 1 RT 1 FCU 0.025
END DEFINE MATERIAL
```

You will create a material definition for a lightweight concrete to use for all members.


2. In the **Material - Whole Structure** dialog **Isotropic** tab, click **Create**.

The **Isotropic Material** dialog opens.
3. Input the required, user-defined material values for a lightweight concrete:
   a. Type CONCRETE-LTW in the Title field.
   b. Type 22 in the Young's Modulus (E) field.
   c. Type 0.17 in the Poisson's Ratio (nu) field.
   d. Type 1.9e-008 in the Density field.
   e. Select CONCRETE from Type of Material drop-down list.
   f. Type 0.025 in the Compressive strength (Fcu) field.

Leave the remaining property fields as zero for this tutorial exercise. The shear modulus is calculated based on E and Poisson's ratio and the thermal coefficient and critical damping default to zero. Refer to the TR.26.1 Define Material (on page 2501) for details.

**Tip:** The material definition can be edited later by selecting the material in the isotropic materials list and clicking Edit.

The new material is added to the Material table and the list of isotropic materials.

**T.2 Specifying member properties**

Next you will assign cross section properties for the beams and columns.
The STAAD input file commands generated are:

```
MEMB PROP
1 4 PRIS YD 300 ZD 275
2 5 PRIS YD 350 ZD 275
3 PRIS YD 350
...
MATERIAL CONCRETE-LTW ALL
```

1. Select the **Properties** page in the Analytical Modeling page control bar.

   ![Properties - Whole Structure dialog](image)

   **Figure 40:**

2. Either:

   - on the **Specifications** ribbon tab, click the **Prismatic** tool in the **Beam Profiles** group

   ![Prismatic tool](image)

   or

   - click **Define**
3. Define a 30 cm x 27.5 cm rectangular, concrete beam:
   a. Select the Rectangle tab.
   b. Check the Material option and select CONCRETE-LTW from the list.
      
      **Note:** This will automatically assign the user-defined material created in a previous procedure to the sections.
   c. Type 300 (mm) in the YD field.
   d. Type 275 (mm) in the ZD field.
   e. Click Add.

4. Repeat step 3 to define the beam member property, this time with a depth (YD) of 350 and a width (ZD) of 275.

5. Define a 35 cm round, concrete column:
   a. Select the Circle tab.
b. Check the **Material** option and select **CONCRETE-LTW** from the list.
c. Specify the diameter (YD) as **350 mm**.
d. Click **Add**.

6. Click **Close**.

7. Assign the 30 cm x 27.5 cm shape to members 1 and 4 (end columns):
   a. Select **Rect 0.30x0.28** in the list of sections.
   b. Select the **Use Cursor to Assign** option in the Assignment Method group.
   c. Click **Assign**.

   The mouse pointer changes to `I`

   d. In the view window, click on members 1 and 4.
   e. To stop assigning members, either:

   click **Assigning**

   or

   press the `<Esc>` key.

8. Repeat step 7 to assign the 0.35cm x 27.5 cm rectangular shape to members 2 and 5 (beams).

9. Repeat step 7 to assign the 0.35cm circular shape to members 3 (center column).
T.2 Specifying geometric constants

You need to orient member 4 so that its longer edges (sides parallel to local Y axis) are parallel to the global Z axis. This requires applying a beta angle of 90 degrees.

In the absence of any explicit instructions, STAAD.Pro will orient the beams and columns of the structure in a predefined way. Orientation refers to the directions along which the width and depth of the cross section are aligned with respect to the global axis system. The rules which dictate this default orientation are explained in General Engineering Theory (on page 2294).

The STAAD input file commands generated are:

```
BETA 90 MEMB 4
```

1. Select the Properties page in the Analytical Modeling page control bar.
   The Properties dialog opens.

2. On the Properties dialog, select the Beta Angle tab.

3. Click Create Beta Angle.
   The Beta Angle dialog opens.
4. Type 90 in the **Angle in Degrees** field.
5. Click **OK**.
   The property **Beta 90** is added to the beta angles list.
6. Select the property **Beta 90**.
7. Assign the beta angle to member 4:
   a. Select the **Use Cursor to Assign** option in the Assignment Method group.
   b. Click **Assign**.
      The mouse pointer changes to ↣
   c. In the view window, click on member 4.
   d. To stop assigning members, either:
      click **Assigning**
      or
      press the <Esc> key.
8. To view the orientation of the member local axes:
   right-click in the view window and select **Labels** from the pop-up menu, check the option for **Beam Orientation**, and click **Apply**.
   or
   press <Shift+O>.
Alternatively, you can assign a beta angle to a selected member set. On the Specification ribbon tab, select the Beam > Beta Angle tool in the Specifications group. Type the beta Angle in Degrees to use. Make sure that the To Selection option is selected and then click OK.

T.2 Specifying Supports

The base nodes of all the columns are restrained against translation and rotation about all the 3 global axes (i.e., fixed supports at those nodes).

The STAAD input file commands generated are:

```
SUPPORTS
1 4 5 FIXED
```

1. On the Select ribbon tab, select the Node Cursor tool in the Cursors group.

2. In the View window, hold <Ctrl> and click on nodes 1, 4, and 5 to select them.
3. In the Supports dialog, click Create.

   The Create Support dialog opens.
4. Select the **Fixed** tab (selected by default) and then click **Assign.**
5. Click any open space in the View window to deselect all selected nodes.

This prevents accidental assignment of unwanted data to those nodes.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL + S>`.

T.2 Specifying Loads

Five load cases are to be created for this structure. Details of the individual cases are explained at the beginning of this tutorial.

The STAAD input file commands generated are:

```
UNIT METER KG
LOAD 1 DEAD LOAD
SELFWEIGHT Y -1
MEMBER LOAD
  2 5 UNI GY -400
LOAD 2 LIVE LOAD
MEMBER LOAD
  2 5 UNI GY -600
LOAD 3 WIND LOAD
MEMBER LOAD
  1 UNI GX 300
  4 UNI GX 500
LOAD 4 DEAD + LIVE
REPEAT LOAD
  1 1.2 2 1.5
LOAD 5 DEAD + WIND
```
T.2 Creating Load Cases 1, 2, and 3

The load values are listed in the beginning of this tutorial in kg and meter units. Rather than convert those values to the current input units, we will conform to those units.

STAAD.Pro does not allow to change the units while editing load cases. An error message is displayed if this is attempted.

Create the primary load cases for this model.

1. Change the force unit to **Kilogram** and the length unit to **Meter**.
   
   Refer to [T.2 Changing the input units of length](on page 466) for details.

2. On the **Loading** ribbon tab, select the **Primary Load Case** in the **Loading Specifications** group.

   ![Create New Primary Load Cases dialog](image)

   The **Create New Primary Load Cases** dialog opens.

3. Enter the properties for the first load case:
   
   a. Type **DEAD LOAD** in the **Title** field.
      
      Leave the **Number** as the default value of 1.

   **Note:** The **Loading Type** list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program’s automatically generating load combinations in accordance with those...
codes. Note that there is a check box labeled *Reducible per UBC/IBC*. This feature is active only when the load case is assigned a Loading Type called *Live* when you create that load case.

Since this tutorial does not use the automatic load combination generation feature, leave the **Loading Type** as **None**.

b. Click **Add**.
c. Click **Close**.

The load case appears under the **Load Cases Details** option in the **Load & Definition** dialog.

4. To create the selfweight dead load, either:

   on the **Loading** ribbon tab, select the **Load Items** tool
   a. Select the **1: DEAD LOAD** entry.

   ![Load & Definition dialog](image)

   **Figure 44:**

   b. Click **Add**.

   The **Add New Load Items** dialog opens with entries for adding load items to the selected load case.

5. Define the selfweight load:

   a. In the **Add New Load Items** dialog, select the **Selfweight Load** option under the **Selfweight** item.
   b. Select **Y** as the **Direction**.
   c. Type **-1.0** in the **Factor** field (multiplier for the calculated selfweight of members).
   d. Click **Add**.
6. With the Add New Load Items dialog still open, add the member loads to be applied to the beams:
   a. In the Add New Load Items dialog, select the Uniform Force tab under the Member Load item.
   b. Select GY as the Direction.

   **Tip:** For these members, since the local Y axis coincides with the global Y axis, selecting Y (local member Y) and GY (global Y direction) have the same effect. Press <Shift+O> to view the member orientation.

   c. Type -400 (kg/m) in the W1 field (magnitude of the uniform force).
      The negative value indicates the load acts "down".
      Leave the offset fields (d1 through d3) as zero.
   d. Click Add.
   e. Click Close.
7. Assign the selfweight load to all members:
   a. Select the **SELFWEIGHT Y -1** entry in the **Load & Definitions** dialog.

   ![Figure 46:](image)

   **Figure 46:**

   b. Select the **Assign to View** option.
   c. Click **Assign**.

   A message box opens prompting you to confirm you want to make this assignment.

   d. Click **Yes**.
   e. Click anywhere in the View window away from the members to deselect all members.

8. Assign the member load to members 2 and 5:
   a. Select the **UNI GY -400 kg/m** entry in the **Load & Definitions** dialog.
b. Press and hold <Ctrl> and then select members 2 and 5 in the View window.

   The **Beams Cursor** tool is automatically selected when the **Load & Definition** dialog is opened.

c. Select the **Assign to Selected Beams** option.

d. Click **Assign**.

   A message box opens to open prompting you to confirm you want to make this assignment.

e. Click **Yes**.

![Figure 48: Structure after uniform force is assigned to beams](image)

**Tip**: Use a scale of 6 kN/m per m for distributed forces if the loads are not visible or are too large for the View window when viewing the entire structure.

9. Repeat steps 3, 4, 6, and 8 to create a new load case titled **Live Load** which contains a uniform force of -600 kg/m in the global Y direction and is applied to members 2 and 5.

   **Note**: Be sure to select the **Live Load** on the **Loading** ribbon tab **Display** group to add load items to this load case.

10. Repeat steps 3 through 4 and then 6 through 8 to create a new load case titled **Wind Load** which contains the following two load items:

   **Note**: Be sure to select the **Wind Load** on the **Loading** ribbon tab **Display** group to add load items to this load case.

   a uniform force of +300 kg/m in the GX (global X) direction and is applied to member 1
   a uniform force of +500 kg/m in the GX (global X) direction and is applied to member 4
T.2 Analysis and Design

The following procedures describe how to add analysis and design commands to the STAAD input file.

**Note:** The analysis design commands are added using either the analytical user interface or the STAAD.Pro Editor. The STAAD.Pro Physical Modeler interface is not used to assign analysis and design commands.

T.2 Creating Load Cases 4 and 5

Use repeat load cases for the load combinations used in a nonlinear analysis.

Load cases 4 and 5 are to be generated using a combination load type called REPEAT LOAD (as opposed to a standard load combination). The instructions at the beginning of this tutorial specify using a PDelta analysis. A PDelta analysis is a non-linear type of analysis. In STAAD, to accurately account for the PDelta effects arising from the simultaneous action of previously defined horizontal and vertical loads, those previous cases must be included as components of the combination case using the REPEAT LOAD type.

1. On the **Loading** ribbon tab, select the **Primary Load Case** in the **Loading Specifications** group.

   ![Create New Primary Load Cases dialog](image)

   The **Create New Primary Load Cases** dialog opens.

---

*Figure 49: Structure after the wind load items are assigned*
2. Enter the properties for the load case:
   a. Type DEAD + LIVE in the Title field.
      Leave the Number as the default value of 4.

      **Note:** The Loading Type list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program's automatically generating load combinations in accordance with those codes. Note that there is a check box labeled Reducible per UBC/IBC. This feature is active only when the load case is assigned a Loading Type called Live when you create that load case.

   b. Click Add.
      The load case appears under the Load Cases Details option in the Load & Definition dialog.

3. To create the dead + live repeat load case:
   a. Select the 4: DEAD + LIVE entry.
Figure 50:

b. Click Add.

The **Add New Load Items** dialog opens with entries for adding load items to the selected load case.

4. Define the repeat load:
   a. In the **Add New Load Items** dialog, select the **Repeat Load** option under the **Repeat Load** item.
   b. Select **1: DEAD LOAD** as the **Available Load Cases** list and then click [>].
      This load is added as Load Case 1 in the **Repeat Load Definition** list.
   c. Type **1.2** in the **Factor** field for **Load Case 1** in the **Repeat Load Definition** list.
      This indicates that the load data values from load case 1 are multiplied by a factor of 1.2, and the resulting values are utilized in load case 4.
   d. Select **2: LIVE LOAD** as the **Available Load Cases** list and then click [>].
      This load is added as Load Case 2 in the **Repeat Load Definition** list.
   e. Type **1.5** in the **Factor** field for **Load Case 1** in the **Repeat Load Definition** list.
   f. Click Add.
Figure 51: Structure with repeat load case selected
5. Click Close.
6. Repeat steps 3 and 5 to create another repeat load case for load case 5, except:

   - title load case 5 **Dead + Wind**
   - add the Dead Load case with a factor of 1.1
   - add the Wind Load case with a factor of 1.3

**Note:** No member assignment is not necessary for repeat load cases. The members assigned to which the primary load cases are assigned are used for the repetition of the load.

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL> +<S>`.

### T.2 Specifying the analysis type

The analysis type for this structure is called P-Delta.

Since this problem involves concrete beam and column design per the ACI code, second-order analysis is required and has to be done on factored loads acting simultaneously. The factored loads have been created earlier as cases 4 and 5. Now is the time to specify the analysis type.

The STAAD input file commands generated are:

```
PDELTA ANALYSIS SMALLDELTA
```

1. On the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group.

The **Analysis/Print Commands** dialog opens.

**Tip:** If the **Analysis/Print Commands** dialog does not open automatically, then click **Define Commands** in the **Analysis - Whole Structure** dialog.

2. Select the **PDelta Analysis** tab.
Note: Leave the parameters on this page as their default values and options.

3. Click Add.
4. Click Close.

Tip: Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL+S>.

T.2 Short-listing the load cases to be used in concrete design

The concrete design should only be performed for load cases 4 and 5 since those are the factored cases. Use the load list to instruct the program to use just these cases.

The STAAD input file commands generated are:

```
LOAD LIST 4 5
```

1. On the Analysis and Design ribbon tab, select the Load List tool in the Analysis Data group. The Load List dialog opens.
2. Press and hold the <Ctrl> key and then select load cases 4: DEAD + LIVE and 5: DEAD + WIND.
3. Click [>].
   Load cases 4 and 5 will be selected and placed in the Load List selection box.
4. Click OK.

T.2 Specifying concrete design parameters

Concrete design parameters allow you to directly specify some of the terms used in the equations for concrete member design.

For example, the grade of concrete or the maximum size of reinforcing bar you want to use.

Note: This tutorial uses the 2011 edition of the American Concrete Institute 318 specification for design (ACI 318-11). Details on the code parameters can be found in D1.F.3 Design Parameters (on page 1481).

The STAAD input file commands generated are:

```
UNIT MMS NEWTON
CODE ACI
CLB 30 ALL
CLS 25 ALL
CLT 25 ALL
FC 25 ALL
FYMAIN 415 ALL
TRACK 1 ALL
```

1. Set the force units as Newton and the length units as Millimeter.
   Refer to T.2 Changing the input units of length (on page 466) for details.
2. On the Analysis and Design ribbon tab, select Concrete in the Design Commands gallery.
The Concrete Design - Whole Structure dialog opens.

3. On the Concrete Design - Whole Structure dialog, select ACI 318 2011 from the Current Code drop-down list.

4. Click Define Parameters in the Concrete Design dialog.

The Design Parameters dialog opens.

5. Specify the clear cover on bottom:
   a. Select the CLB parameter tab.
   b. Type as 30 (mm) in the parameter value field.
   c. Click Add.

6. Repeat step 5 to define the remaining parameters:

Figure 54:
### Parameter

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CLS</td>
<td>25</td>
</tr>
<tr>
<td>CLT</td>
<td>25</td>
</tr>
<tr>
<td>FC</td>
<td>25</td>
</tr>
<tr>
<td>FYMAIN</td>
<td>415</td>
</tr>
<tr>
<td>TRACK</td>
<td>1 (select option from list)</td>
</tr>
</tbody>
</table>

7. Click **Close**.

The **Concrete Design** dialog displays the parameters in the concrete design command.
Figure 55:
8. Assign these parameters to all the members in the model:
   a. Select the first unassigned parameter in the **Concrete Design | Whole Structure** dialog.
   b. Select the **Assign to View** option.
   c. Click **Assign**.
      
      A message dialog opens confirming you want to make this assignment.
   d. Click **Yes**.
   e. Repeat steps 8a though 8d for each unassigned parameter to assign them to all members.

T.2 Specifying design commands

Design commands are the actual instructions for the design of beams and columns.

The STAAD input file commands generated are:

```
DESIGN BEAM 2 5
DESIGN COLUMN 1 3 4
```

1. On the **Concrete Design - Whole Structure** dialog, click **Commands**.

   The **Design Commands** dialog opens.

   ![Design Commands dialog]

2. Select the **Design Beam** tab and then click **Add**.
3. Select the **Design Column** option and then click **Add**.
4. Click **Close**.
5. Assign the Design Beam command to the beams:
   a. Press and hold the <Ctrl> key and click members 2 and 5

   **Tip:** The Beams Cursor tool is selected when the Concrete Design - Whole Model dialog opens.

   b. Select the Assign to Selected Beams option.
   c. Click Assign.

   A message box opens prompting you to confirm you want to make this assignment.
   d. Click Yes.

6. Repeat steps 5 to assign the Design Column command to members 1, 3, and 4.

   **Tip:** Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL +S>.

T.2 Viewing the input command file

You can inspect the text data file created during this tutorial.

1. On the Utilities ribbon tab, select the Command File tool in the Edit group.

   The STAAD Editor window opens.
2. (Optional) You modify the data of the structure in this editor if necessary.
3. Select **File > Exit Editor** in the STAAD.Pro Editor window to close.

As stated in "**T.2 Methods of creating the model** (on page 449)", you could also have created the same model by typing the relevant STAAD commands into a text file using the STAAD.Pro Editor. If you would like to understand that method, proceed to the next section. If you want to skip that part, proceed to "**T.2 Performing the analysis and design** (on page 499)" where you will perform the analysis and design on this model.

**T.2 Creating the model using the command file**

As an alternatively to the procedures described in the proceeding tutorial, you can also create the same STAAD input file using the STAAD.Pro Editor.

**Note:** A STAAD input file is a plain text file that uses the .std file extension. Therefore any standard text editor such as Notepad can also be used to create the command file. However, the STAAD.Pro Editor offers the advantage of syntax checking as you type the commands. The STAAD command syntax are highlighted by command, keyword, value, etc.
To start a new STAAD input file using the STAAD.Pro Editor, follow the procedure described in "Creating a new structure" (on page 451), except select the Open STAAD Editor check box in step 7. The STAAD.Pro Editor window opens with the basic commands for your model entered.

For this tutorial, delete all the command lines displayed in the editor window and type the lines shown below. While not necessary, this will allow you to learn more about the required and optional command lines for an input file.

STAAD commands are not case sensitive (i.e., they may be typed in upper or lower case letters). By convention, this and most input files use all caps, though.

For most all commands and keywords, the first three letters of a keyword are all that are needed. The rest of the letters of the word are not required, but are useful to present a user-friendly command language in mostly plain English for later reference. By convention, the required letters in a command or keyword are underlined here ("PLANE" = "PLA" = "plane" = "pla").

**STAAD SPACE RC FRAMED STRUCTURE**

Every input has to start with the word STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNIT METER KN
```
Specifies the unit to be used.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0 0 0 ; 2 0 3.5 0 ; 3 6 3.5 0</td>
</tr>
<tr>
<td>4 6 0 0 ; 5 6 0 6 ; 6 6 3.5 6</td>
</tr>
</tbody>
</table>

Joint number followed by X, Y and Z coordinates are provided above. Semicolon signs (;) are used as line separators. That enables you to provide multiple sets of data on one line.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 ; 2 2 3 ; 3 3 4</td>
</tr>
<tr>
<td>4 5 6 ; 5 6 3</td>
</tr>
</tbody>
</table>

Defines the members by the joints they are connected to.

<table>
<thead>
<tr>
<th>UNIT MMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER PROPERTY AMERICAN</td>
</tr>
<tr>
<td>1 4 PRIS YD 300 ZD 275</td>
</tr>
<tr>
<td>2 5 PRIS YD 350 ZD 275</td>
</tr>
<tr>
<td>3 PRIS YD 350</td>
</tr>
</tbody>
</table>

Member properties have been defined above using the PRISMATIC attribute for which YD (depth) and ZD (width) values are provided in MM unit. When YD and ZD are provided together, STAAD considers the section to be rectangular. When YD alone is specified, the section is considered to be circular. Details are available in Section 5 of the Technical Reference Manual.

<table>
<thead>
<tr>
<th>CONSTANTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>E 22 MEMB 1 TO 5</td>
</tr>
</tbody>
</table>

Material constant E (modulus of elasticity) is specified as 22KN/sq.mm following the command CONSTANTS.

<table>
<thead>
<tr>
<th>UNIT METER KN</th>
</tr>
</thead>
<tbody>
<tr>
<td>CONSTANTS</td>
</tr>
<tr>
<td>DENSITY 25.0 ALL</td>
</tr>
<tr>
<td>POISSON 0.17 ALL</td>
</tr>
</tbody>
</table>

Length unit is changed from MMS to METER to facilitate the input of Density. Next, the Poisson’s Ratio is specified.

<table>
<thead>
<tr>
<th>BETA 90 MEMB 4</th>
</tr>
</thead>
</table>

In the absence of any explicit instructions, STAAD will orient the beams and columns of the structure in a predefined way (see Section 1 of the Technical Reference Manual for details.) In order to orient member 4 so that its longer edges (sides parallel to local Y axis) are parallel to the global Z axis, you must apply a beta angle of 90 degrees.

<table>
<thead>
<tr>
<th>SUPPORT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 4 5 FIXED</td>
</tr>
</tbody>
</table>

Joints 1, 4 and 5 are defined as fixed supported.

<table>
<thead>
<tr>
<th>UNIT METER KG</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD 1 DEAD LOAD</td>
</tr>
</tbody>
</table>

Force units are changed from KN to KG to facilitate the input of loads. Load case 1 is initiated along with an accompanying title.

| SELFWEIGHT Y -1 |
One of the components of load case 1 is the selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

```
MEMBER LOAD
2 5 UNI
GY -400
```

Load 1 contains member loads also. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load. Loads are applied on members 2 and 5.

```
LOAD 2 LIVE LOAD
```

Load case 2 is initiated along with an accompanying title.

```
MEMBER LOAD
2 5 UNI
GY -600
```

Load 2 also contains member loads. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load. Loads are applied on members 2 and 5.

```
LOAD 3 WIND LOAD
```

Load case 3 is initiated along with an accompanying title.

```
MEMBER LOAD
1 UNI
GX 300
4 UNI
GX 500
```

Load 3 also contains member loads. GX indicates that the load is in the global X direction. The word UNI stands for uniformly distributed load. Loads are applied on members 1 and 4.

```
LOAD 4 DEAD + LIVE
```

Load case 4 is initiated along with an accompanying title.

```
REPEAT LOAD
1 1.2 2 1.5
```

Load case 4 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases specified earlier. This repeat load case instructs the program to analyze the structure for loads from cases 1 and 2 acting simultaneously. The load data values from load case 1 are multiplied by a factor of 1.2, and the resulting values are utilized in load case 4. Similarly, the load data values from load case 2 are multiplied by a factor of 1.5, and the resulting values too are utilized in load case 4.

```
LOAD 5 DEAD + WIND
```

Load case 5 is initiated along with an accompanying title.

```
REPEAT LOAD
1 1.1 3 1.3
```

This repeat load case instructs the program to analyze the structure for loads from cases 1 and 3 acting simultaneously.

```
PDELTA ANALYSIS
```

The PDELTA ANALYSIS command is an instruction to the program to execute a second-order analysis and account for P-delta effects.

```
LOAD LIST 4 5
```
The above LOAD LIST command is a means of stating that all further calculations should be based on the results of load cases 4 and 5 only. The intent here is to restrict concrete design calculations to that for load cases 4 and 5 only.

```
START CONCRETE DESIGN
CODE ACI
UNIT MMS
NEWTON
CLT 25 ALL
CLB 30 ALL
CLS 25 ALL
FC 25 ALL
FYMAIN 415 ALL
TRACK 1 ALL
```

The first line is the command that initiates the concrete design operation. The values for the concrete design parameters are defined in the above commands. Design is performed per the ACI Code. The length units are changed from METER to MMS to facilitate the input of the design parameters. Similarly, force units are changed from KG to NEWTON. The TRACK value dictates the extent of design related information which should be produced by the program in the output. The parameters specified include CLT (Clear cover for top surface), CLB (Clear cover for bottom surface), CLS (Clear cover for sides), FC(Strength of concrete), and FYMAIN (Ultimate strength of steel). These parameters are described in Section 3 of the Technical Reference Manual.

```
DESIGN BEAM 2 5
DESIGN COLUMN 1 3 4
```

The above commands instruct the program to design beams 2 and 5 for flexure, shear and torsion, and to design columns 1, 3 and 4 for axial load and biaxial bending.

```
END CONCRETE DESIGN
```

This command terminates the concrete design operation.

```
FINISH
```

This command terminates the STAAD run.

**Save** the input file and close the editor. The model is opened in the STAAD.Pro interface.

### T.2 Performing the analysis and design

STAAD.Pro performs Analysis and Design simultaneously.

1. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.

   ![Run Analysis tool](image.png)

   As the analysis progresses, several messages appear on the screen as shown in the figure below.
2. Select the View Output File option once the analysis and design are complete.

The three options are indicative of what will happen after you click Done.

**View Output File**
This option opens the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands specified during the model generation process. It also provides you with important messages of any errors that were encountered, and if so, whether the analysis and design was successfully completed or not. See T.1 Viewing the output file (on page 433) for details on viewing and understanding the contents of the output file.

**Go to Post Processing Mode**
This option opens the graphical Post-processor mode, which can be used to extensively review and verify the results. This mode allows you to view the results graphically, plot result diagrams, produce reports, etc. See T.1 Post-Processing (on page 441) for details on the Post processing mode.

**Stay in Modeling Mode**
This option closes the dialog and remains in the Model generation mode of the program, where you initiated the analysis. This is useful if you want to make further changes to the input file.

3. Click Done. The STAAD.Pro Output Viewer window opens.
T.2 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly.

For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

**Note:**
If you did not select to open the output file after running the analysis in the previous procedure, you can open it any time through the user interface. On the Utilities ribbon tab, select the Analysis Output tool in the View group.

**Tip:** By default, the output file contains a listing of the entire input also. You may choose not to print the echo of the input commands in the Output file. On the Analysis and Design ribbon tab, select the Miscellaneous Commands > Set Echo tool option from the menu bar and the select the Echo Off option in the Set Echo dialog.

It is *strongly recommended* that you review the entire output file to ensure that the results are reasonable and that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the post-processing mode. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.
15. POISSON 0.17
16. DENSITY 1.9E-008
17. TYPE CONCRETE
18. TYPE CONCRETE
19. TYPE CONCRETE
20. STRENGTH FCU 0.025
21. END DEFINE MATERIAL
22. MEMBER PROPERTY AMERICAN
23. 1 4 PRIS YD 300 ZD 275
24. 2 5 PRIS YD 350 ZD 275
25. 3 PRIS YD 350
26. CONSTANTS
27. BETA 90 MEMB 4
28. MATERIAL CONCRETE-LTW ALL
29. SUPPORTS
30. 1 4 5 FIXED
31. UNIT METER KG
32. LOAD 1 LOADTYPE NONE TITLE DEAD LOAD
33. SELFWEIGHT Y -1 LIST ALL
34. MEMBER LOAD
35. 2 5 UNI GY -400
36. LOAD 2 LOADTYPE NONE TITLE LIVE LOAD
37. MEMBER LOAD
38. 2 5 UNI GY -600
39. STAAD SPACE
40. LOAD 3 LOADTYPE NONE TITLE WIND LOAD
41. MEMBER LOAD
42. 1 UNI GX 300
43. 4 UNI GX 500
44. LOAD 4 LOADTYPE NONE TITLE DEAD + LIVE
45. REPEAT LOAD
46. 1 1.2 2 1.5
47. LOAD 5 LOADTYPE NONE TITLE DEAD + WIND
48. REPEAT LOAD
49. 1 1.1 3 1.3
50. PDELTA ANALYSIS SMALLDELTA

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     5, TOTAL DEGREES OF FREEDOM =      18
TOTAL LOAD COMBINATION CASES =     0 SO FAR.
++ Adjusting Displacements.
++ Adjusting Displacements.
51. START CONCRETE DESIGN

CONCRETE DESIGN
52. CODE ACI
53. UNIT MMS NEWTON
54. CLB 30 ALL
55. CLS 25 ALL
56. CLT 25 ALL
57. FC 25 ALL
58. FYMAIN 415 ALL
59. TRACK 1 ALL
## Design Beam 2/5

### Design Summary

<table>
<thead>
<tr>
<th>Status</th>
<th>Pass</th>
<th>Type</th>
<th>Beam</th>
<th>Length: 6000.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>1.000</td>
<td>Criteria:</td>
<td>Torsion</td>
<td></td>
</tr>
<tr>
<td>Critical Clause</td>
<td>9.5.3/9.5.4</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Cross Section

| Shape: Rectangular | Width: 275.00 | Depth: 350.00 |

### Longitudinal Bar Layout

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars: Nums</th>
<th>Size: 13M</th>
<th>Location Start</th>
<th>Location End</th>
<th>Distance From Face: 31.41</th>
<th>Anchor: Yes Yes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right</td>
<td>3</td>
<td></td>
<td>0.00</td>
<td>6000.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Top</td>
<td>3</td>
<td></td>
<td>0.00</td>
<td>6000.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bottom</td>
<td>2</td>
<td>13M</td>
<td>0.00</td>
<td>6000.00</td>
<td>36.41</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>1</td>
<td>13M</td>
<td>982.60</td>
<td>5017.40</td>
<td>36.41</td>
<td>No No</td>
</tr>
<tr>
<td>Left</td>
<td>3</td>
<td></td>
<td>0.00</td>
<td>6000.00</td>
<td>31.41</td>
<td>Yes Yes</td>
</tr>
</tbody>
</table>

### Transverse Bar Layout

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir.</th>
<th>From</th>
<th>To</th>
<th>Reqd.</th>
<th>Prov.</th>
<th>Nums</th>
<th>Size</th>
<th>Spacing</th>
<th>Legs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>6000.00</td>
<td>31.96</td>
<td>51.35</td>
<td>48</td>
<td>13M</td>
<td>127.66</td>
<td>2</td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>6000.00</td>
<td>31.96</td>
<td>51.35</td>
<td>48</td>
<td>13M</td>
<td>127.66</td>
<td>2</td>
</tr>
</tbody>
</table>

---

### DesignBeam 5

### Design Summary

<table>
<thead>
<tr>
<th>Status</th>
<th>Pass</th>
<th>Type</th>
<th>Beam</th>
<th>Length: 6000.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>1.000</td>
<td>Criteria:</td>
<td>Torsion</td>
<td></td>
</tr>
<tr>
<td>Critical Clause</td>
<td>9.5.3/9.5.4</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Cross Section

| Shape: Rectangular | Width: 275.00 | Depth: 350.00 |

### Longitudinal Bar Layout

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars: Nums</th>
<th>Size: 13M</th>
<th>Location Start</th>
<th>Location End</th>
<th>Distance From Face: 31.41</th>
<th>Anchor: Yes Yes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right</td>
<td>3</td>
<td>13M</td>
<td>0.00</td>
<td>6000.00</td>
<td>31.41</td>
<td>Yes Yes</td>
</tr>
</tbody>
</table>
## TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir.</th>
<th>From</th>
<th>To</th>
<th>Asv</th>
<th>Reqd.</th>
<th>Prov.</th>
<th>Nums</th>
<th>Size</th>
<th>Spacing</th>
<th>Legs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.01</td>
<td>1500.00</td>
<td>35.14</td>
<td>50.42</td>
<td>13</td>
<td>13M</td>
<td>125.00</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.01</td>
<td>1500.00</td>
<td>35.14</td>
<td>50.42</td>
<td>13</td>
<td>13M</td>
<td>125.00</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Y</td>
<td>1500.00</td>
<td>4500.00</td>
<td>31.96</td>
<td>52.44</td>
<td>25</td>
<td>13M</td>
<td>125.00</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Z</td>
<td>1500.00</td>
<td>4500.00</td>
<td>31.96</td>
<td>52.44</td>
<td>25</td>
<td>13M</td>
<td>125.00</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Y</td>
<td>4500.00</td>
<td>5999.99</td>
<td>35.14</td>
<td>50.42</td>
<td>13</td>
<td>13M</td>
<td>125.00</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Z</td>
<td>4500.00</td>
<td>5999.99</td>
<td>35.14</td>
<td>50.42</td>
<td>13</td>
<td>13M</td>
<td>125.00</td>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

---

### Member: 5 Design Ends

61. DESIGN COLUMN 1 3 4

**STAAD SPACE**

-- PAGE NO. 6

**STAAD.PRO CONCRETE DESIGN - (ACI-318-14) v2.0**

********************************************************************************

**Units: NEWTON, MMS**

(Unless Noted Otherwise)

**Member:** 1

**DESIGN SUMMARY**

<table>
<thead>
<tr>
<th>Status</th>
<th>Critical Ratio</th>
<th>Critical Clause</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pass</td>
<td>0.609</td>
<td>10.5.2</td>
</tr>
</tbody>
</table>

**CROSS SECTION**

- **Shape:** Rectangular
- **Width:** 275.00
- **Depth:** 300.00

### LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Nums</th>
<th>Size</th>
<th>Start</th>
<th>End</th>
<th>Distance From Face</th>
<th>Anchor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>3</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>31.41</td>
<td>Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>3</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>36.41</td>
<td>Yes</td>
</tr>
<tr>
<td>Left</td>
<td>1</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>31.41</td>
<td>Yes</td>
</tr>
<tr>
<td>Right</td>
<td>1</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>31.41</td>
<td>Yes</td>
</tr>
</tbody>
</table>

---

**Member: 1 Design Ends**

**STAAD SPACE**

-- PAGE NO. 8

**STAAD.PRO CONCRETE DESIGN - (ACI-318-14) v2.0**

********************************************************************************

**Units: NEWTON, MMS**

(Unless Noted Otherwise)
### DESIGN SUMMARY

<table>
<thead>
<tr>
<th>Status</th>
<th>Pass</th>
<th>Type</th>
<th>Column</th>
<th>Length: 3500.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>0.782</td>
<td>Criteria: Flexure</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Critical Clause: 10.5.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CROSS SECTION

| Shape: Circular | Dia: 350.00 |

### LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars</th>
<th>Size</th>
<th>Start</th>
<th>Distance</th>
<th>Anchor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Evenly Dist.</td>
<td>8</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>Yes</td>
</tr>
</tbody>
</table>

### TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir.</th>
<th>From</th>
<th>To</th>
<th>Reqd.</th>
<th>Prov.</th>
<th>Nums</th>
<th>Size</th>
<th>Spacing</th>
<th>Legs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>3500.00</td>
<td>31.96</td>
<td>33.71</td>
<td>19</td>
<td>13M</td>
<td>194.44</td>
<td>2</td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>3500.00</td>
<td>31.96</td>
<td>33.71</td>
<td>19</td>
<td>13M</td>
<td>194.44</td>
<td>2</td>
</tr>
</tbody>
</table>

---

**STAAD SPACE**  
-- PAGE NO. 9

**STAAD.PRO CONCRETE DESIGN - (ACI-318-14) v2.0**  
******************************************************************************

Units: NEWTON, MMS  
(Unless Noted Otherwise)

Member : 3

### DESIGN SUMMARY

<table>
<thead>
<tr>
<th>Status</th>
<th>Pass</th>
<th>Type</th>
<th>Column</th>
<th>Length: 3500.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>1.000</td>
<td>Criteria: Torsion</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Critical Clause: 10.5.3/10.5.4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CROSS SECTION

| Shape: Rectangular | Width: 275.00 | Depth: 300.00 |

### LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars</th>
<th>Size</th>
<th>Start</th>
<th>Distance</th>
<th>Anchor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>3</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>3</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Left</td>
<td>1</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Right</td>
<td>1</td>
<td>13M</td>
<td>0.00</td>
<td>3500.00</td>
<td>Yes</td>
</tr>
</tbody>
</table>

### TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir.</th>
<th>From</th>
<th>To</th>
<th>Reqd.</th>
<th>Prov.</th>
<th>Nums</th>
<th>Size</th>
<th>Spacing</th>
<th>Legs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>3500.00</td>
<td>31.96</td>
<td>55.79</td>
<td>31</td>
<td>13M</td>
<td>116.67</td>
<td>2</td>
</tr>
</tbody>
</table>
T.2 Post-Processing

STAAD.Pro offers extensive result verification and visualization facilities. These facilities are accessed from the Post Processing Mode. The Post Processing mode is used to verify the analysis and design results and generate reports.

T.2 Opening the postprocessing workflow

You can open the Postprocessing workflow anytime there are current analysis results for your model.

If you selected to open the postprocessing mode after T.2 Performing the analysis and design (on page 499), then you can skip to step 2.

1. On the Workflows panel, select Postprocessing.

The Results Setup dialog opens.
2. (Optional) Select the load cases for which to display the results.

**Tip:** All load cases are selected by default.

For this tutorial, we will use all load cases.

### T.2 Viewing the deflection diagram

1. Select the **Displacement** page in **Postprocessing** page control bar.

**Tip:** This page is the default page when you open the Postprocessing workflow.

**Note:** You can display the Deflection diagram on most results pages by selecting the **Deflection** tool. The tool remains depressed when the displacement is displayed.

The displacement diagram is drawn on the structure. The **Node Displacements table** and **Beam Relative Displacement Detail** table open. See **T.2 The Node Displacements Table** (on page 512) for details on these tables.

2. (Optional) If the deflection is not exaggerated enough to clearly identify, you can change the scale:
   a. On the Results ribbon tab, select the **Scale** tool in the **Configuration** group.

The **Diagrams** dialog opens displaying the **Scales** tab.
Tip: You can also open the **Diagrams** dialog by selecting the **Structure** tool or by right-clicking in the View area and then selecting **Structure Diagrams** from the pop-up menu.

b. Change the **Displacement** value.

**Tip:** Smaller numbers exaggerate the deflected shape.

Type a value of 5 (mm per m) to clearly see the deflection due to load case 5.

c. Click **Apply**.

**Tip:** You can set the **Apply Immediately** option on this tab to view the scale change as you use the up/down arrows in the Displacement field.

d. Click **OK**.

3. To change the load case being displayed, select from the current **Load** list in the window status bar.

```
Tip: Alternatively, select the **Structure** tool ![Structure Tool] in the **Configuration** group on the **Results** toolbar. The active load case can be selected on the **Load and Results** tab in the **Diagrams** dialog.
```

The results displayed are for the selected load case only.

Select **5: DEAD + WIND** for this tutorial.
4. Annotate the deflection at specific nodes:
   a. On the Results ribbon tab, select the **Annotate** tool in the **Configuration** group.

   The **Annotation** dialog opens.

   b. On the **Ranges** tab, select the **Ranges** option and then type 2 3 6 (spaced node number list) in the **Nodes** field.
c. Select the **Node** tab and then check the **Resultant** option.

Resultant stands for the square root of sum of squares of values of X, Y and Z displacements.

d. Click **Annotate** and then click **Close**.

The structure deflection diagram is annotated for load case 5, as in the following figure.
5. Change the display units for displacement:

   The units in which displacement values are displayed in the post-processing mode are referred to as the display units.

   a. On the File ribbon tab, select Display Options on the Settings tab.

      The Options dialog opens.

   b. Select the Structure Units tab.

   c. Change Displacement selection from mm (millimeter) to cm (centimeter).
d. Click OK. The diagram will be updated to reflect the new units.

T.2 The Node Displacements Table

When the Displacements page is selected, two tables open on the right side of the program window.

The Node Displacements table lists the displacement values for every node for every selected load case. The Beam Relative Displacement Detail table displays the displacements along beams at intermediate points.

Note: The load cases included in results tables can be restricted using the Results Setup dialog. Refer to T.2 Restricting the load cases for results (on page 522) for details.

Tip:
You can reopen any closed tables from the Tables dialog, which is opened by selecting the Tables tool in the Windows group on the View ribbon tab.
The **Node Displacements** table

<table>
<thead>
<tr>
<th>Node</th>
<th>L/C</th>
<th>X (mm)</th>
<th>Y (mm)</th>
<th>Z (mm)</th>
<th>Resultant (mm)</th>
<th>rX (rad)</th>
<th>rY (rad)</th>
<th>rZ (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1 DEAD LOA</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>2 LIVE LOAD</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>3</td>
<td>3 WIND LOA</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>4</td>
<td>4 DEAD + LIV</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>5</td>
<td>5 DEAD + WI</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>1 DEAD LOA</td>
<td>-0.062</td>
<td>-0.038</td>
<td>0.369</td>
<td>0.376</td>
<td>0.000</td>
<td>0.000</td>
<td>-0.001</td>
</tr>
</tbody>
</table>

*Figure 59:*

**All** This tab presents all nodal displacements in tabular form for all load cases and all degrees of freedom.

**Summary** This tab, shown in the figure below, presents the maximum and minimum nodal displacements (translational and rotational) for each degree of freedom. All nodes and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).
The Beam Relative Displacement Detail table

![Beam Relative Displacement Detail Table](image)

Figure 60:

**All** tab presents the displacements of members at intermediate section points. All specified members and all specified load cases are included. The table shows displacements along the local axes of the members, as well as their resultants.

**Max Displacements** tab presents the summary of maximum sectional displacements (see figure below). This table includes the maximum displacement values and location of its occurrence along the member, for all specified members and all specified load cases. The table also provides the ratio of the span length of the member to the resultant maximum section displacement of the member.

T.2 Viewing the force and moment diagrams

**Tip:** You can display force or moment diagrams on most any page in the Post-Processing, but is recommended to use the Beam | Forces page to do so.

1. Select the **Beam Results** page on the **Postprocessing** page control bar.

   **Note:** You can display force and moment diagrams on most results pages by selecting corresponding tool from the **View Result** group on the **Results** ribbon tab. The corresponding tool remains depressed when the diagram is displayed.

   The bending moment $M_z$ is drawn on the structure by default. The **Beam End Forces** table and **Beam Force Detail** table open. T.2 The **Beam Forces Table** (on page 519) for details on these tables.

2. To change the force or moment (i.e., degree of freedom) diagram displayed, either:

   - select the corresponding diagram tool from the **View Result** group on the **Results** ribbon tab
   - or

   select **Structure** tool in the **Configuration** group on the **Results** ribbon tab and then select the degrees of freedom to display from the **Beam Forces** group on the **Loads and Results** tab.
Table 22: Results diagram tools

<table>
<thead>
<tr>
<th>Tool</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Axial Force Diagram" /></td>
<td>Displays the axial force diagram</td>
</tr>
<tr>
<td><img src="image2" alt="Shear Y Diagram" /></td>
<td>Displays the shear Y diagram</td>
</tr>
<tr>
<td>Tool</td>
<td>What it does</td>
</tr>
<tr>
<td>------</td>
<td>--------------</td>
</tr>
<tr>
<td><img src="92x656" alt="Image" /></td>
<td>Displays the shear Z diagram</td>
</tr>
<tr>
<td><img src="92x616" alt="Image" /></td>
<td>Displays the torsion diagram</td>
</tr>
<tr>
<td><img src="92x577" alt="Image" /></td>
<td>Displays the bending Y diagram</td>
</tr>
<tr>
<td><img src="92x538" alt="Image" /></td>
<td>Displays the bending Z diagram</td>
</tr>
<tr>
<td><img src="92x499" alt="Image" /></td>
<td>Displays the beam stress (tension and compression) diagrams.</td>
</tr>
</tbody>
</table>

**Note:** The **Load and Results** tab on the **Diagrams** dialog can also be used to select the color for each diagram as well as the display method (i.e., hatched, filled, or outline).

**Tip:** You can display multiple diagrams simultaneously.

3.  (Optional) If the force or moment diagram is not exaggerated enough to clearly identify, you can change the scale:
   a. On the Results ribbon tab, select the **Scale** tool in the **Configuration** group.

   The **Diagrams** dialog opens displaying the **Scales** tab.

   **Tip:** You can also open the **Diagrams** dialog by selecting the **Structure** tool or by right-clicking in the View area and then selecting **Structure Diagrams** from the pop-up menu.

   b. Change the value in the corresponding **Results Scales** group to the diagram currently displayed.

   **Tip:** Smaller numbers exaggerate the diagram.

   Type a value of 50 (kN/m per m) to clearly see the **Bending Z** due to load case 5.

   c. Click **Apply**.

   **Tip:** You can set the **Apply Immediately** option on this tab to view the scale change as you use the up/down arrows in any of the **Result Scales** fields.

   d. Click **OK**.

4. To change the load case being displayed, select from the current **Load** list in the window status bar.
Tip: Alternatively, select the **Structure** tool in the **Configuration** group on the **Results** toolbar. The active load case can be selected on the **Load and Results** tab in the **Diagrams** dialog.

The results displayed are for the selected load case only.

5. Annotate the bending Z moment diagram to display the maximum moment in the beams for load case 5:
   a. Select **Results > View Value**.
      The **Annotation** dialog opens.
   b. On the **Ranges** tab, select the **Ranges** option and then type 2 5 (spaced node number list) in the **Beams** field.
   c. Select the **Beam Results** tab and then check the **Maximum** option in the **Bending** group.
d. Click **Annotate** and then click **Close**.

The bending Z moment diagram is annotated for load case 5, as in the following figure.

![Bending Z Moment Diagram](image)

*Figure 61:*

6. Change the display units for bending moments:
The units in which results values are displayed in the post-processing mode are referred to as the display units.

a. On the File ribbon tab, select Display Options on the Settings tab.

   The Options dialog opens.

b. Select the Force Units tab.

c. Change Moment selection from kNm (kilonewton meters) to kip-ft (kilopound feet).

d. Click OK.

   The diagram updates to reflect the new units.

T.2 The Beam Forces Table

When the Beam Results page is selected in the Postprocessing workflow, two tables open on the right side of the program window.

The Beam End Forces table lists the axial forces and shear forces, bending and torsional moments in all selected beams for all selected load cases are displayed in a tabular form along the right half of the screen.

The Beam Force Detail table lists force and moment values for every node for every selected load case.

Note: The load cases included in results tables can be restricted using the Results Setup dialog. Refer to T.2 Restricting the load cases for results (on page 522) for details.
Tip:
You can reopen any closed tables from the Tables dialog, which is opened by selecting the Tables tool in the Windows group on the View ribbon tab.

The Beam End Forces table

Figure 62:

All  This tab presents all forces and moments corresponding to all 6 degrees of freedom at the start and end of each selected member for all selected load cases.

Summary  This tab, shown in the next figure, presents the maximum and minimum values (forces and moments) for each degree of freedom. All beams and all Load Cases specified during the Results
Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).

Envelope  This tab shows a table consisting of the maximum and minimum for each degree of freedom for each member, and the load case responsible for each of those values.

T.2 Viewing the force and moment graphs

The Graphs results in the Postprocessing workflow are used to view moment and force graphs such as Axial, Bending zz, Shear yy and Combined Stresses for individual members.

1. Display the beam results graphs:
   a. On the Results ribbon tab, select the Tables tool in the View Results group.
   b. In the Beam Results group, select Graphs.
   The View window shows the loading on the structure. On the right side of the screen, the force/moment diagrams appear.

2. Select a member in the View window.
   The graphs are plotted for that member in the data area.

   Figure 63: Bending, shear, and axial graphs for Beam 2 under load case 5: DEAD + WIND.

3. To change the diagrams displayed for the current beam and load case selection:
   a. Right-click on any graph window and select Diagrams from the pop-up menu.
      The Diagram dialog opens.
   b. Set the check box for the degrees of freedom you wish to view in the diagram.
c. Click OK.
The selected degree of freedom is plotted in that window.

T.2 Restricting the load cases for results
To restrict the load cases for which results are viewed, use the following procedure.

1. On the Results ribbon tab, select the Select Load Case tool in the Configuration group.

![Diagram](image.png)

Figure 64:
The Results Setup dialog opens.
2. Click [<<].
All load cases are moved from the Selected list to the Available list.
3. Select two load cases to use for viewing results:
   a. Press and hold the `<Ctrl>` key and then select load cases 1 DEAD LOAD and 3 WIND LOAD.
   b. Click `[>].`
   c. Click OK.

T.2 Using Member Query

Member query is a facility where several results for specific members can be viewed at the same time from a single dialog. It is also a place from where many of the member attributes such as the property definition, specifications (releases, truss, cable, etc.) and beta angle can be changed for input purposes.

**Note:** The Beam Cursor tool is active when the Beam Results page is selected.

1. Double-click a member in the View window.

   **Tip:** Alternately, if you have selected a member you can then select Tools > Query > Member.

   For this example, double-click on member 4.

   The Beam dialog opens.

   In the Modeling mode, you can actively edit members using this dialog. In the Post-Processing mode, the member data is presented for review only.
**Note**: The tabs in the member query dialog reflect the properties of the member along with any analysis and design results available. Changing the model will void any results and thus remove these tabs until another analysis is performed.

**Figure 66:**

2. Select the **Shear Bending** tab.

Here you can review shear and bending diagrams for the member for a selected load case. A table presents the shear and bending at specific locations along the member length. The slider control can be used to review the values at distance.
3. Select the **Deflection** tab.

Here, a similar diagram of the member deflection is shown for the selected load case. A table displays the deflection values at specific points along the member length.
4. Select the **Concrete Design** tab.

**Note:** This tab is only available for concrete members which have been designed using the concrete design command and after a successful analysis has been performed.
5. (Optional) Click Print

The query results for the current dialog tab are printed using the report output settings. Refer to T.2 Creating Customized Reports (on page 530) for details on the report settings.

6. Click Close.

To review the member query results of another beam, the Beam dialog must be closed. Repeat this procedure for another member to query it.

T.2 Producing an on-screen report

Occasionally, we will come across a need to obtain results conforming to certain restrictions, such as, say, the resultant node displacements for a few selected nodes, for a few selected load cases, sorted in the order from low to high, with the values reported in a tabular form. The facility which enables us to obtain such customized on-screen results is the Report menu on top of the screen.

Here, you will create a report that includes a table with the member major axis moment (MZ) values sorted in the order High to Low, for members 1 and 4 for all the load cases.
1. Press and hold <Ctrl> and click both members 1 and 4 in the View window to select them.

2. On the Results ribbon tab, select the Reports > Beam End Forces tool in the Reports group.

   ![Beam End Force dialog](image)

   The Beam End Force dialog opens.

3. Specify the report contents:
   a. Select the Sorting tab.
   b. Select the Moment-Z option in the Sort By End Force group.
   c. Select List from High to Low option in the Set Sorting Order group.
   d. Check the Absolute Values option in the If Sorting done group.

4. Select the Loading tab and ensure all the five load cases are added to the Selected list.

5. (Optional) You can save this report for future use,
   a. Select the Report tab
   b. Type a Title for the report.
   c. Set the Save Report check box.

6. Click OK.

   The member end forces sorted table opens with the MZ values sorted from High to Low based on Absolute numbers.
T.2 Taking Pictures

To take a screen capture of the View window contents for use generating external reports, use the following procedure.

1. On the Utilities ribbon tab, select the Take Picture tool in the Utilities group.

![Take Picture tool]

The Picture # dialog opens.

![Picture # dialog]

2. (Optional) Type a Picture ID

   The ID is incremented automatically.

3. (Optional) Type a Caption

   **Note:** For most View window contents, the caption will be automatically completed with a description of the contents.

4. Click OK.

   This picture is saved for use in reports.
T.2 Creating Customized Reports

STAAD.Pro offers extensive report generation facilities.

Items which can be incorporated into such reports include input information, numerical results, steel design results, etc. You can choose from among a select set of load cases, mode shapes, structural elements, etc. You can also include any picture of the screen taken using the Take Picture tool. Other customizable parameters include the font size, title block, headers, footers, etc.

1. Open the Report Setup:
   a. Select the File ribbon tab.
      The Backstage view opens.
   b. Select the Report tab and then Setup.
      The Report Setup dialog opens.

   ![Report Setup Dialog]

   **Note:** The Available items list is filtered by the drop-down selection list above it. This allows you to sort your reports.

2. Add model Output to the report:

   **Tip:** Job Information from the Input is selected by default.

   a. Select Output from the drop-down list.
      The Available list displays output items.
   b. Double-click Node Displacement Summary in the Available list.
      This item is added to the Selected list.
c. Double-click **Beam Max Moments** in the Available list.
   This item is added to the Selected list.

d. Select **Pictures** from the drop-down list.
   The Available list displays the picture captured earlier in the tutorial.

e. Double-click **Picture 1**.
   This item is added to the Selected list.

Leave the **Report Detail Increments** at 10. This is the number of segments into which a member is divided for sectional reports (i.e., displacements, forces, etc.).

3. Select the load cases to include in the report as well as how to order the results:
   a. Select the **Load Cases** tab.
   b. If all the load cases are not already included in the output, click [>>].
   c. Select the **by Load Case** option for the **Grouping for Load Tables**.
   d. Select the **by Node/Beam** option for **Grouping for Result Tables**.

![Report Setup](image)

4. (Optional) Select the **Picture Album**

   **Tip:** You can manage pictures taken using the **Take Picture** tool here.
5. Customize the title block of your report with your organization’s information:
   a. Select the **Name and Logo** tab.
   b. Click in the empty text field and type the name and address of your organization.
   c. (Optional) Click **File Graphic**
      
      By default, the Bentley Systems, Inc. logo is used for STAAD.Pro reports.
      
      Logo files must be in Bitmap (file extension .bmp) format.
   d. Click **Font** in the **Text** group.
      
      The Font dialog opens. You can select any Windows font, style, and size. Click OK when done.
6. Click **OK**.

   **Tip:** You can click **Print** to print the report directly from this dialog, but it is recommended to preview the report prior to printing.

7. Select the **File** ribbon tab and then select **Report > Print Preview**.

   The Print Preview window opens with the report contents. Here you can review and print the report. The first and the last pages of the report are shown in the next two figures.
Tutorials
T.2 - RC Framed Structure

Job Information

Structure Type: SPACE FRAME

Number of Nodes: 5
Number of Element: 5
Number of Basic Load Cases: 3
Number of Combination Load Cases: 0

Included in this printout are results for load cases:

<table>
<thead>
<tr>
<th>Type</th>
<th>LC</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary</td>
<td>1</td>
<td>DEAD LOAD</td>
</tr>
<tr>
<td>Primary</td>
<td>2</td>
<td>LIVE LOAD</td>
</tr>
<tr>
<td>Primary</td>
<td>3</td>
<td>WIND LOAD</td>
</tr>
<tr>
<td>Primary</td>
<td>4</td>
<td>DEAD + LIVE</td>
</tr>
<tr>
<td>Primary</td>
<td>5</td>
<td>DEAD + WIND</td>
</tr>
</tbody>
</table>

Node Displacement Summary

<table>
<thead>
<tr>
<th>Node</th>
<th>LC</th>
<th>X (mm)</th>
<th>Y (mm)</th>
<th>Z (mm)</th>
<th>Resultant (mm)</th>
<th>dx (rad)</th>
<th>dy (rad)</th>
<th>dz (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max X</td>
<td>0</td>
<td>6.798</td>
<td>-0.042</td>
<td>-0.007</td>
<td>0.798</td>
<td>0.001</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Min X</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Max Y</td>
<td>0</td>
<td>5</td>
<td>0</td>
<td>0</td>
<td>5</td>
<td>0.001</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Min Y</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Max Z</td>
<td>0</td>
<td>1.159</td>
<td>0.069</td>
<td>0.325</td>
<td>1.159</td>
<td>0.001</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Min Z</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Max dx</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Min dx</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Max dy</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Min dy</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Max dz</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Min dz</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
8. Click **Close**.

This brings us to the end of this tutorial.

**T.3 - Analysis of a slab**

This tutorial provides step-by-step instructions for modeling and analysis of a slab supported along two edges.
T.3 Methods of creating the model

As explained in T.1 Methods of creating the model (on page 373) in Tutorial 1, there are three methods of creating the structure data:

1. Using the STAAD.Pro Physical Modeler interface (on page 539),
2. using the traditional STAAD.Pro analytical modeling interface (on page 547),
3. and using the command file (on page 587).

All three methods of creating the model are explained in this tutorial. The physical modeling method is explained first, beginning with “T.3 Creating the Model using the Physical Modeler (on page 539),” then the analytical model method is explained beginning with “T.3 Description of the tutorial problem (on page 536).” The command file method is explained in “T.3 Creating the model using the command file (on page 587).”

T.3 Description of the tutorial problem

The structure for this project is a slab fixed along two edges. You will model it using 6 quadrilateral (i.e., 4-noded) plate elements. The structure and the mathematical model are shown in the figures below. It is subjected to selfweight, pressure loads and temperature loads. The purpose of this tutorial to create the model, assign all required input, perform the analysis, and review the results.
Basic Data for the Structure

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element properties</td>
<td>Slab is 300 mm thick</td>
</tr>
</tbody>
</table>
### Attribute | Data
--- | ---
Material Constants | E, Density, Poisson, Alpha — Default values for concrete
Supports | Nodes along 2 adjacent edges; fixed against rotation and translation
Primary Loads | Load 1: Selfweight  
Load 2: Pressure Load of 300 kg/m² acting vertically downwards  
Load 3: 75 degree F uniform expansion, plus top surface is 60 degrees hotter than the bottom
Combination Loads | Case 101: Case 1 + Case 2  
Case 102: Case 1 + Case 3
Analysis Type | Linear Elastic

### T.3 Creating a new structure

On the Start page **New** tab, you will provide some initial data necessary for building the model.

1. On the Start page, select **New**.
   - The **New** page opens to the **Model Info** tab.
2. Type as **Plates Tutorial** in the **File Name** field.
3. Specify a **Location** where the STAAD input file will be located on your computer or network.
   - You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.
4. Select **Physical** for the **Type** of model.
   - This option selects the modeling method you want to use:
     - **Analytical** For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.
     - **Physical** For creating a model using the STAAD.Pro Physical Modeler interface.
     - **Building** For creating a building model structure using the Building workflow.
5. Select **Metric** as the system of **Units**.
   - **Tip:** The units can be changed later if necessary, at any stage of the model creation.
6. (Optional) Select the **Job Info** tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.
7. Click **Create**.
The STAAD.Pro modeling environment opens and your model file is then opened in the STAAD.Pro Physical Modeler.

Alternatively, if you want to use either the analytical modeling workflow or the command input file editor to create the model, choose Analytical, click Create, and then proceed to either "T.3 Creating the model using the analytical user interface (on page 547)" or "T.3 Creating the model using the analytical user interface (on page 547)," respectively.

T.3 Creating the Model using the Physical Modeler

You are now ready to start building the model geometry. The steps for doing this are described in the following sections.

Tip: Refer to the STAAD.Pro Physical Modeler Application Window Layout for reference on the application window.

T.3 Generate the model geometry

1. On the Model ribbon tab, select the Grid tool in the Create group.

The Create Grid dialog opens.

2. Supply the grid details:
   a. Type Grid 1 in the Name field.
   b. Select XZ as the Plane.
   c. Leave the Creation Method as By Spacing.
   d. Type 3 in the Number of Spaces in X and 2 m in the Grid spacing X field.
   e. Type 2 in the Number of Spaces in Z and 2 m in the Grid spacing Z field.
   f. Click OK.
      The grid is created.

3. Orient the view from the top:
   a. (Optional) If the rotation controls are hidden, click the rotation widget in the lower-right corner of the view window.
   b. Select the Switch to top view tool.
   c. (Optional) Click the rotation widget again to hide the controls.

   Tip: Alternatively, you can press and hold the right mouse button and drag the mouse pointer to quickly rotate the view so it is easy to select the column bases.

4. On the Model ribbon tab, select the Surface tool in the Create group.
5. Draw the surface:
   a. Click on the grid point at (0,0,0) (i.e., top-left corner).
   b. Click on the grid corners in a counterclockwise order. Double-click on the last point to stop drawing surface vertices.
      (6,0,0)
      (6,0,4)
      (0,0,4)
   c. Select the Surface tool again to deactivate the tool.

6. On the View ribbon tab, select the Local Axis tool in the Model group.

![Local Axis](image)

The local axes are displayed on the center of the surface. Note that local axis x is aligned with the edge defined by the first two nodes.

### T.3 Specifying element and material properties

1. Select the surface element.

2. On the Surface Tools ribbon tab, select the Material tool in the Assign Properties group.

![Assign Material](image)

The Assign material dialog opens.

3. Assign a catalog concrete material:
   a. Select the Source as Catalog.
   b. Select the Type as Standard.
   c. Select the Country as United States.
   d. Select the Material type as CONCRETE_ACI318.
   e. Select 4000 psi from the Material name list.
   f. Click OK

The dialog closes and the material name is displayed on the surface.


![Assign Thickness](image)

The Assign Surface Thickness dialog opens.

5. Type 300 mm in the Thickness field and then click OK.

The dialog closes and the thickness is displayed on the surface.
T.3 Specifying supports

**Tip:** You may want to turn off the display of Loads to make selecting the surface easier. To do so, select the **Loads** tool in the **Toggle Attributes** group on the **View** ribbon tab.

1. Select the surface and the nodes along the “top” edge:
   a. Click in an empty area of the view window to deselect all objects.
   b. Click on the surface to select it.
   c. While holding `<Ctrl>`, drag a rectangular area surrounding nodes N2 and N3.
2. On the **Surface Tools** ribbon tab, select the **Linear** tool in the **Mat and Edge Supports** group.

   The **Assign Custom Linear Support** dialog opens.

3. Specify the support parameters:
   Leave the **Support type** selection as **Restraints**.
   a. Type 3 in the **Segments** field.
   b. Check all the restrained options for X, Y, and Z in both the **Translation** and **Rotation** groups.
   c. Click **OK**.

   A series of blocks are drawn along the edge to indicate the support condition.

4. Repeat steps 1 through 3 but select the “left” edge nodes (N1 and N2) and use 2 segments.
T.3 Specifying load groups 1 and 2

1. On the Spreadsheet ribbon tab, select the Load Cases tool in the References group.

The Load Cases spreadsheet opens.

2. For Load 1:
   a. Type Dead Load in the description field of Load case 1.
   b. Select Dead as the Type.
   c. Type -1 in the Self-weight multiplier in field Y.

   Note: This value is negative.

   d. Press <Enter>. 
3. Create a load group for the external pressure load:
   a. On the Model ribbon tab, select the Load Case tool in the Load group.

   The Add Load Case dialog opens.
   b. Type External Pressure Load in the Name field.
   c. Select Floor Unspecified as the Type.
   d. Click OK.

   The new load group is created and selected as the active load group.

4. On the Surface Tools ribbon tab, select the Pressure tool in the Loads group.

   The Add Surface Uniform Pressure dialog opens.

5. Specify the pressure load:
   a. Type -300 kg/m² in the Pressure field.
   b. Select Global Y as the Direction.

   Tip: Since the surface is parallel to the XZ plane, you could also have selected local 3 as to achieve the same direction. However, due to the counterclockwise selection of nodes, the local 3 axis is oriented in the opposite direction of the Global Y. Therefore, you would use a positive magnitude for that case.
   c. Click OK.

   The temperature load must be added directly to the analytical model in the STAAD.Pro interface. This is done after you generate the analytical model.

T.3 Generate the Analysis Model

You have now completed the majority of the modeling portion of this tutorial. You will now send the model data back to STAAD.Pro where you will add the final load case, create load combinations, add output commands, select analysis criteria, and specify post-processing print commands.

1. Change the mesh size:
a. Select the **File** ribbon tab. 
The backstage view opens.

b. Select the **Options** tab. 
The **Options** dialog opens.

c. Select the **Analysis model** tab.

d. Type 2.0 m in the **Maximum allowed distance between nodes** field.

e. Click **OK**.

2. Either:

On the **Model** ribbon tab, select the **Return to Analytical Modeling** tool in the **STAAD.Pro** group

![Image of the Return to Analytical Modeling dialog]

or

On the **File** ribbon tab, select **Create analysis model** in the backstage tabs
The Return to Analytical Modeling dialog opens and displays the progress of the analytical model being generated.

T.3 Create primary load cases

1. In STAAD.Pro, select the **Loading** page in the **Analytical Modeling** page control bar.

   The **Loading** ribbon tab is selected and the **Load & Definition** dialog opens.

2. On the **Loading** ribbon tab, select the **Primary Load Case** tool in the **Loading Specifications** group.

   ![Image of the Add New: Load Cases dialog]

   The **Add New: Load Cases** dialog opens.
3. Create a pair of primary load cases:
   a. Type Dead Load as the Title for Load Case 1.
      Leave the Number as the default (1) and leave the Loading Type as None.
   b. Click Add.
      Note that the dialog stays open but the load case number is automatically incremented.
   c. Type External Pressure Load as the Title.
      Again, it is not necessary to select a Loading Type.
   d. Click Add.
   e. Click Close.

4. Assign the reference loads to the primary load cases:
   a. On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group.

   The Add New Load Items dialog opens.
   b. Select the Repeat Load > Reference Load tab on the left.
   c. Select R1: "Load group 1" in the Available Load Cases list.
   d. Click >.
      The reference load case is added to the selected loads list.
   e. Click Add.
   f. Click Close.
      The reference load case is now added included in the primary load case for analysis.

5. In the window status bar, select 2: External Pressure Load as the current load case.
6. Repeat Step 4 to add the reference load for Load group 2 to the External Pressure Load primary load case.
**T.3 Add temperature load case**

1. Create a primary load case titled **Temperature Load** and then select it in the status bar.
   a. On the **Loading** ribbon tab, select the **Primary Load Case** tool in the **Loading Specifications** group. 

   The **Add New Load Cases** dialog opens.

   b. Type **Temperature Load** as the **Title**.
   
   There are no load combinations used in this model, so you can leave the Loading Type as None.

   c. Click **Add**.

   d. Click **Close**.

2. In the program status bar, select **3: Temperature Load** as the current load case.

3. To generate and assign the third load type:
   a. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   The **Add New Load Items** dialog opens.

   b. Select the **Temperature** option under the **Temperature Loads** item.

   c. Type **40** in the **Temperature Change for Axial Elongation** field.

   d. Type **30** in the **Temperature Differential from Top to Bottom** field.

   Leave the **Temperature Differential from Side to Side (Local Z)** field as **0** (default).

   e. Click **Add** and then click **Close**.
4. Assign the temperature load case to all of the elements:
   a. In the Load & Definition dialog, select TEMP 40 30.
      This is the input command for the temperature load.
   b. Select the Assign to View option.
   c. Click Assign.
      A message dialog opens confirming you want to make this assignment.
   d. Click Yes.

T.3 Creating the model using the analytical user interface

The following procedures describe how to use the traditional STAAD.Pro analytical user interface to build the model. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

**Tip:** Refer to GS. Application Window Layout (on page 50) for reference on the application window.

T.3 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc.
The STAAD input file commands generated are:

```
JOINT  COORDINATES
1 0 0 0 ; 2 2 0 0 ; 3 2 0 2 ; 4 0 0 2
5 4 0 0 ; 6 4 0 2 ; 7 6 0 0 ; 8 6 0 2
9 2 0 4 ; 10 0 0 4 ; 11 4 0 4 ; 12 6 0 4
ELEMENT  INCIDENCES  SHELL
1 1 2 3 4 ; 2 2 5 6 3 ; 3 5 7 8 6
4 4 3 9 10 ; 5 3 6 11 9 ; 6 6 8 12 11
```

In this tutorial, you will optionally explore four different methods to create the model:

**T.3 Creating the Plates - Method 1**

To create the plate elements for the slab using a mixture of drawing an element and the Copy/Paste facility, do the following steps.

**T.3 Setup the Grid**

1. On the **Geometry** ribbon tab, select the **Snap Grid/Plate (Quad)** tool in the **Structure** group.
   ![Structure Wizard](image)

   The **Snap Node/Beam** dialog closes and the **Snap Node/Plate** dialog opens.

2. On the **Snap Node/Plate** dialog, click **Create**.
   A dialog opens which will enable us to set up a grid.
   Within this dialog, there is a drop-down list from which you can select Linear, Radial, or Irregular form of grid lines.

   - **Linear** used to place the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board
   - **Radial** used to place construction lines to appear in a spider-web style, which makes it is easy to create circular type models where members are modeled as piece-wise linear straight line segments
   - **Irregular** used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane

3. Select **Linear**, which is the Default Grid.
   In our structure, the segment consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane.
   So, in this dialog, let us keep **X-Y** as the **Plane** of the grid. The size of the model that can be drawn at any time is controlled by the number of **Construction Lines** to the left and right of the origin of axes, and the **Spacing** between adjacent construction lines.

4. Type a **Name** of **Grid 1**.
5. Select X-Z as the Plane option.
6. Type 6 as the number of lines to the Right of the origin along X and 4 above the origin along Z.
   Leave the spacing as 1 m.
7. Click OK.
8. In the Snap Node/Beam grids list, check the new Grid 1 option.
   You can create any number of grids. By providing a name, each new grid can be identified for future reference.
   
   **Tip:** Please note that these settings are only used to generated construction lines. These construction lines enable you to easily draw the structure but do not restrict our overall model to those limits.

   **Tip:** To change the settings of this grid, select the name in the Snap Node/Beam dialog and then click Edit.

   The check by Default Grid is automatically cleared. STAAD.Pro will only display a single grid at a time.
**T.3 Create Plates Method 1 Element 1**

1. In the View window, click at the origin (0, 0) to create the first node. A line is “rubber-banded” between this node and the mouse pointer, which previews the edge of the plate to be placed with the next mouse click.
2. Click on the following points to create nodes and automatically join successive nodes by beam members.

(2, 0, 0)
(2, 0, 2)
(0, 0, 2)

3. Click **Close** in the **Snap Node/Plate** dialog.

4. Press **<Shift+P>** to display the plate number.
It is very important that we save our work often, to avoid loss of data and protect our investment of time and effort against power interruptions, system problems, or other unforeseen events.

**Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing `<CTRL+S>`.

**T.3 Create Plates Method 1 Element 2**

Creating Elements by Coping and Pasting

Examining the structure shown in “**T.3 Description of the tutorial problem** (on page 536)” of this tutorial, it can be seen that the remaining elements can be easily generated copying the existing plate and then pasting the copied element at specific distances.

1. On the **Geometry** ribbon tab, select the **Plates Cursor** tool in the **Selection** group.

2. Select plate 1 in the View window.

3. Either:
   - Right-click and select **Copy** from the pop-up menu
   - or
   - on the **Geometry** ribbon tab, select the **Copy** tool in the **Clipboard** group
   - or
   - press `<Ctrl+C>`
4. To paste the copied plate element, either:
   Right-click and select **Paste** from the pop-up menu
   or
   on the **Geometry** ribbon tab, select the **Paste** tool in the **Clipboard** group
   or
   press <Ctrl+V>
   The **Paste with Move** dialog opens.

5. Select the **By the following X, Y and Z values** option and then type 2 (m) in the X field.
   Leave the Y and Z values as 0.
6. Click **OK**.
   The dialog closes and the plate is copied to Plate 2 at the specified X axis increment.
7. Repeat steps 4 through 6 except with an X of 4 (m) to create plate 3 from plate 1.

You have now created one half of the symmetric set of plates. You can now copy and paste all three plates to complete the structure.

8. To select all plates, either:
   click-and-drag a window around all three plates using the **Plates Cursor** tool
   or
   press **<Ctrl+A>**
   or
on the Geometry ribbon tab, select the Plates Cursor > All Plates tool in the Selection group.

9. Repeat steps 3 through 6 to copy and paste all the plates, except with a Z of 2 m (leave the X and Y values as 0).

10. Click anywhere in away from the plates to deselect them.

T.3 Create Plates Delete to Try Another Method

If you want to explore the remaining methods of creating this model, the current structure will have to be entirely deleted.

1. To select all plates, either:
   - click-and-drag a window around all three plates using the Plates Cursor tool
   - or
   - press <Ctrl+A>
   - or
   - on the Geometry ribbon tab, select the Plates Cursor > All Plates tool in the Selection group.

2. Either:
   - press <Delete>
   - or
   - on the Plate Tools ribbon tab, select the Delete tool in the Clipboard group
   
   A message dialog opens to confirm the deletion of the selected plates.

3. Click OK
   A message dialog opens indicating that orphan nodes have been created and to confirm their deletion.

4. Click Yes.
   The entire structure is now deleted.
T.3 Creating the Plates - Method 2

To create the plate elements using a mixture of drawing an element and the Translational Repeat facility, use the following steps.

**T.3 Method 2 Creating Element 1**

In this method, you will use the Translational Repeat feature in STAAD.Pro to create the model. First, you need at least one existing entity to use as the basis for the translational repeat.

This method uses the same drawing grid from Method 1. Refer to “T.3 Setup the Grid (on page 548).”

1. In the View window, click at the origin (0, 0) to create the first node.
   A line is “rubber-banded” between this node and the mouse pointer, which previews the edge of the plate to placed with the next mouse click.

2. Click on the following points to create nodes and automatically join successive nodes by beam members.
   
   (2, 0, 0)
   (2, 0, 2)
   (0, 0, 2)
3. Click Close in the Snap Node/Beam dialog.
   The grid is hidden.
4. Press <Shift+P> to display the plate number.

**T.3 Method 2 Creating Elements 2 3**

In Method 1, it required two separate executions of the Copy/Paste function to create elements 2 and 3. That is because, that feature does not contain a provision for specifying the number of copies you want to create. However, with Translational Repeat you can specify the number of copies and distances to each.

1. On the Geometry ribbon tab, select the Plates Cursor tool in the Selection group.

2. Select plate 1 in the View window.
3. On the Geometry ribbon tab, select the Translational Repeat tool in the Structure group.

The Translational Repeat dialog opens.
4. Specify the translation repeat parameters:
   a. Select X as the **Global Direction** option.
   b. Type 2 (m) in the **Default Step Spacing** field.
   c. Type 2 in the **No of Steps** field and press the Tab key (or click the up arrow to increment the value to 2).

   The Step and Spacing table updates to reflect two steps each at 2m spacing.

   Leave the **Renumber Bay** and **Link Steps** options unchecked and the **Generation Flags** set to **All**.

**Note:**

Using the **All** option for Generation Flags (default), all loads, properties, design parameters, member releases, etc. on the selected entities will automatically be copied along with the entities. You can limit this to geometry only or to geometry and properties only. In our example, it does not matter because no other attributes have been assigned yet.

**Renumber Bay** is used to specify a custom number scheme for created entities, instead of the sequential number that the program otherwise generates. The **Linked Steps** option is used to generate linking members between new entities that have some distance separating them from the original entities.

5. Click **OK**

Elements 2 and 3 are created using the translational repeat parameters.
6. To select all plates, either:
   click-and-drag a window around all three plates using the Plates Cursor tool
   or
   press <Ctrl+A>
   or
   on the Geometry ribbon tab, select the Plates Cursor > All Plates tool in the Selection group.

7. Repeat steps 3 and 4 except select Z as the Global Direction and the No of Steps as 1.
   Tip: Be sure to type 2 (m) in the Default Step Spacing field

8. Click anywhere in away from the plates to deselect them.
T.3 Creating the Plates - Method 3

To create the plate elements using the **Structure Wizard** program, use the following steps.

A program installed with STAAD.Pro called **Structure Wizard** offers a library of pre-defined structure templates (also referred to as “prototypes”), such as Pratt Truss, Northlight Truss, Cylindrical Frame, etc. A surface entity such as a slab or wall, which can be defined using 3-noded or 4-noded plate elements, is one such template. You can also create your own library of structure prototypes. From this wizard, a structural model may parametrically be generated, and can then be incorporated into your main structure.

1. On the **Geometry** ribbon tab, select the **Structure Wizard** tool in the **Structure** group.

The **Structure Wizard** window opens.

![Structure Wizard window](image)

*Figure 81: The Structure Wizard window*

**Tip:** The **Open Structure Wizard** option in the **Where do you want to go?** dialog in the beginning stage of creating a new structure also opens this application.

2. The unit of length should be specified prior to the generation of a model.
   a. Select **File > Select Units** in the Structure Wizard window.

   The **Select Units** dialog opens.
b. Select **Meters**.
c. Click **OK**.

3. From the **Model Type** drop-down list, select **Surface/Plate Models**.

4. Either:
   - double-click on the **Quad Plate** option
   - drag the **Quad Plate** option to the right side of the **Structure Wizard** window

   The Select Meshing Parameters dialog opens.
5. Specify the meshing boundary corners and the individual element data as follows:

   a. Enter the following **Corners** data:

<table>
<thead>
<tr>
<th>Corner</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>B</td>
<td>6</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>C</td>
<td>6</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>D</td>
<td>0</td>
<td>0</td>
<td>4</td>
</tr>
</tbody>
</table>

   The Length values are automatically calculated.

   b. Type 3 and the **Divisions** along AB and CD sides and 2 for the **Divisions** along BC and DA sides.

   Leave the **Bias** values all as 1.

   c. Click **Apply**.

   The mesh is generated.

   **Tip:** If you made a mistake, right-click in the drawing area and select **Change Property** from the pop-up menu. The dialog opens and you can edit the mesh generations parameters.
6. Select **File > Merge Model with STAAD.Pro Model.**
   You are prompted to confirm that you want to transfer the model data with your STAAD.Pro model.

7. Click **Yes.**

   The **Paste with Move** dialog opens.

8. Click **OK.**

   In this tutorial, there is no existing model nor is there a need to shift the model from the origin.

   The model will now be transferred to the main window.

   ![Plates Tutorial.std - Whole Structure](image)

   If you want to proceed with assigning the remainder of the data, go to "[T.3 Changing the input units of length](#)" (on page 566).

---

### T.3 Creating the Plates - Method 4

To create the plates using the Mesh Generation facility, use the following steps.

This method uses the same drawing grid from Method 1. Refer to "[T.3 Setup the Grid](#)" (on page 548)."

The STAAD.Pro GUI contains a facility for generating a mesh of elements from a boundary (or superelement) defined by a set of corner nodes. The boundary must form a closed surface and must lie within a plane, though that plane can be inclined to any of the global planes.

You define the boundary by selecting the corner nodes. If these nodes do not exist, they must be created before they can be selected.

1. Create the nodes:
   a. On the **Snap Node/Plate** dialog, click **Snap Node/Plate.**
   b. Click at the four corners of the grid at \((0, 0, 0)\), \((6, 0, 0)\), \((6, 0, 4)\), and \((0, 0, 4)\).

   **Tip:** If the node points are not visible, press `<SHIFT+K>`.

2. In the **Snap Node/Plate** dialog, click **Close.**
3. Use the Plates Cursor and select this plate.
4. To start a quadrilateral mesh of the existing plate element:
a. On the Plate Tools dialog, select the Generate Plate Mesh tool in the Model group.

The Chose Meshing Type dialog opens.

b. Select the Quadrilateral Meshing option.

c. Click OK

The Select Meshing Parameters dialog opens.

5. Complete the meshing parameters:
   a. Type 3 and the Divisions along AB and CD sides and 2 for the Divisions along BC and DA sides.
   b. Click Apply.

The surface is meshed.

6. Re-number the plate elements:
   a. On the Plate Tools ribbon tab, select the Renumber Plates tool in the Model group.
A warning message opens.

**b. Click Yes.**

The **Renumber** dialog opens.

![Renumber Dialog](image)

- **c. Double-click Z Coordinate** in the **Available Sort Criteria** list.
- **d. Double-click X Coordinate** in the **Available Sort Criteria** list.

The Z coordinate entry should appear *above* the X coordinate entry in the **Selected Sort Criteria** list.

- **e. Click Accept.**

The six plates are re-numbered as shown below.

![Plates Tutorial](image)

As an alternative in step 1, you can press and hold `<Ctrl>` to create only disconnected nodes. Then, when you can use the **Generate Mesh > Generate Plate Mesh** tool in the **Plate** group on the **Geometry** ribbon tab to generate the plates from those nodes, rather than re-meshing an existing plate. You then use the plate bound selection cursor ![cursor](image) to select these corner nodes to define a meshed boundary.
**Tip:** Pressing the `<Ctrl>` key while clicking on grid points creates new nodes without connecting those nodes with beams or plates. If the `<Ctrl>` key is not kept pressed, the nodes become connected.

### T.3 Changing the input units of length

To specify element properties for the structure, it is more convenient to use length units of centimeter instead of meter. This requires changing the current length units of input.

The STAAD input file commands generated are:

```
UNIT CM KN
```

1. On the **Geometry** ribbon tab, select the **Input Units** tool in the **Structure** group.

   The **Set Input Units** pop-up opens.

   ![Set Input Units](image)

2. Select **Metric** as the **Input Units**.
3. Set the **Length** units to **Centimeter**.
   - Leave the Force units as KiloNewton.
4. Click **Apply**.

### T.3 Specifying Element Properties

Just as properties are assigned to members, properties must be assigned to plate elements too. The property required for plates is the plate thickness (or the thickness at each node of elements if the slab has a varying thickness).

The STAAD input file commands generated are:

```
ELEMENT PROPERTY
1 TO 6 THICKNESS 30
```
1. To select all plates, either:
   
   click-and-drag a window around all three plates using the **Plates Cursor** tool
   
   or
   
   press **<Ctrl+A>**
   
   or
   
   on the **Geometry** ribbon tab, select the **Plates Cursor > All Plates** tool in the **Selection** group.

2. On the **Specification** ribbon tab, select the **Plate Thickness** tool in the **Plate Profiles** group.

The **Properties - Whole Structure** and the **Plate Element/ Surface Property** dialogs open.
3. Specify the material and thickness:
   a. Type 30 cm for the **Plate Element Thickness** at each corner.
      
      **Tip:** Typing a value in the first field will automatically populate the remaining fields as a uniform thickness is typical.

   b. Ensure that **Material** option is checked and that **CONCRETE** is selected.
      
      This instructs the program to assign the material properties of Concrete (E, Poisson, Density, Alpha, etc.) will be assigned along with the plate thickness.
      
      **Tip:** The material property values so assigned will be the program defaults. You can review those values by clicking **Materials** in the **Properties - Whole Structure** dialog.

   c. Click **Assign**.
   d. Click **Close**.

   The plate property is added to the **Properties - Whole Structure** dialog list and is assigned to the plate elements.
4. Click anywhere in the drawing area to deselect the selected entities.

**Tip:** This is a best practice to follow when using the assignment feature. When an entity is highlighted, clicking on any Assign option is liable to cause an undesired attribute to be assigned to that entity.

**T.3 Specifying Material Constants**

When “T.3 Specifying Element Properties” (on page 566), you used the **Material** option. Consequently, the material definition (E, Density, Poisson’s Ratio, etc.) of concrete were assigned to the plates along with the properties

The STAAD input file commands generated are:

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2171.85
POISSON 0.17
DENSITY 2.35616e-005
ALPHA 1e-005
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 2.7579
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```
**T.3 Specifying Supports**

The slab is fixed-supported along the entire length of two adjacent sides. However, when modeled as plate elements, the supports can be specified only at the nodes along those edges, and not at any point between the nodes.

**Tip:** The finer the mesh (i.e., the larger the number of elements), then the more supported nodes you would be able to model. For this tutorial, the 2 meter mesh is used as a simple demonstration.

The STAAD input file commands generated are:

```
SUPPORTS
1 2 4 5 7 10 FIXED
```

**Note:** The node numbers may be different depending on the method of plate creation used. Regardless, the nodes should be those along the X and Z axis.

1. Press `<Shift+N>` to turn on the display of the **Node Numbers**.
   
   This will assist in the identification of the nodes used as supports.

2. On the **Geometry** ribbon tab, select the **Nodes Cursor** tool in the **Selection** group.

3. While pressing the `<Ctrl>` key, select the nodes on two adjacent edges of the plate group:
   
   These nodes are fixed supports in the model.
   
   For example, in the following figure these nodes are:
   
   1
   7
   4
   5
   8
   2
4. On the **Specification** ribbon tab, select the **Fixed** tool in the **Supports** group.

    The Create Support dialog opens with the **Fixed** tab selected.

5. Click **Assign**.
Note: The Assign button is active because you selected the nodes previously.

Figure 85: The structure with fixed supports along two adjacent edges

The fixed support type is also added to the Supports - Whole Structure dialog as S2.

T.3 Specifying Primary Load Cases

Three primary load cases are required for this structure.

The STAAD input file commands generated are:

```
UNIT METER KG
LOAD 1 DEAD LOAD
SELF Y -1.0
LOAD 2 EXTERNAL PRESSURE LOAD
ELEMENT LOAD
1 TO 6 PR GY -300
LOAD 3 TEMPERATURE LOAD
1 TO 6 TEMP 40 30
```

1. Change the units:

   Note: The pressure load value listed in the beginning of this tutorial is in KN and meter units. Rather than convert that value to the current input units, this tutorial will conform to those units. The current input units, which you last set by specifying the thickness was centimeter. Therefore, you need to change the force unit to Kilogram and the length units to Meter.

   a. Click the drop-down arrow beside the current Input Units in the application window status bar. The Set input Units dialog opens.
   b. Select the length units as Meter and the force units as Kilogram.
   c. Click Apply.
2. Add the dead load case:
   a. On the Loading ribbon tab, select the Primary Load Case tool in the Loading Specifications group.

   The Add New Load Cases dialog opens.

   ![Add New Load Cases dialog](image)

   b. Type Dead Load as the Title for Load Case 1.
      Leave the Number as the default (1) and leave the Loading Type as None.

      **Note:** The Loading Type list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program's automatically generating load combinations in accordance with those codes. Note that there is a check box labeled Reducible per UBC/IBC. This feature is active only when the load case is assigned a Loading Type called Live when you create that load case.

   c. Click Add.

   The newly created load case will now appear under the Load Cases Details in the Load & Definition dialog.
3. Create the selfweight load:
   a. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   The **Add New Load Items** dialog opens.

   ![Add New Load Items dialog](image)

   b. Select the **Selfweight Load** option under the **Selfweight** item.
   c. Select the **Direction** as **Y**
   d. Type the **Factor** as **-1.0**.

   The negative number signifies that the selfweight load acts opposite to the positive direction of the global axis (Y in this case) along which it is applied.

   e. Click **Add**.

4. Assign the selfweight load to all of the elements:
   a. In the **Load & Definition** dialog, select **SELFWEIGHT Y -1**.
   b. Select the **Assign to View** option.
   c. Click **Assign**.

   A message dialog opens confirming you want to make this assignment.

   d. Click **Yes**.

5. Add the pressure load case:
   a. On the **Loading** ribbon tab, select the **Primary Load Case** tool in the **Loading Specifications** group.
The Add New Load Cases dialog opens.

b. Type External Pressure Load as the Title.

Again, there is no need to associate the load case with any code based Loading Type so leave the selection as None.

c. Click Add.

6. On the Loading ribbon tab Display group, select 2: External Pressure Load from the Load drop-down list.

7. Create the pressure load:

   a. On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group.

   The Add New Load Items dialog opens.

   b. Select the Pressure on Full Plate option under the Plate Loads item.

   This type enables the load to be applied on the full area of the element.

   Note: The Concentrated Load is for applying a concentrated force on the element. The Trapezoidal and Hydrostatic options are for defining pressures with intensities varying from one point to another. The Partial Plate Pressure Load is useful if the load is to be applied as a “patch” on a small localized portion of an element.

   c. Type -300 kg/m$^2$ in the W1 field (force)

   d. Select GY as the Direction (global Y direction).

   e. Click Add.

   f. Click Close.
8. Repeat steps 5 and 6 to create a third load case titled Temperature Load and then select it on the Loading ribbon tab Display group.

9. To generate and assign the third load type:
   a. On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group.

   The Add New Load Items dialog opens.
   b. Select the Temperature option under the Temperature Loads item.
   c. Type 40 in the Temperature Change for Axial Elongation field.
   d. Type 30 in the Temperature Differential from Top to Bottom field.

   Leave the Temperature Differential from Side to Side (Local Z) field as 0 (default).
   e. Click Add and then click Close.
Figure 86:

10. To apply the pressure load and temperature load on all the plates, repeat step 4 for each load item.

Figure 87: The pressure load applied to all plate elements
T.3 Analysis and Design

The following procedures describe how to add analysis and design commands to the STAAD input file.

Note: The analysis design commands are added using either the analytical user interface or the STAAD.Pro Editor. The STAAD.Pro Physical Modeler interface is not used to assign analysis and design commands.

T.3 Creating load combinations

This tutorial requires creating two combination cases.

The STAAD input file commands generated are:

LOAD COMBINATION 101 CASE 1 + CASE 2
1 1.0 2 1.0
LOAD COMBINATION 102 CASE 1 + CASE 3
1 1.0 3 1.0

1. Define load case 4 as a load combination:
   a. On the Loading ribbon tab, select the Combination Load Case tool in the Loading Specifications group.
   
   The Create New Load Combinations dialog opens.
   b. Select the Define Combinations option from the left-hand side.
   c. Type 101 in the Load No: field.
   d. Type Case 1 + Case 2 in the Title field.

   Leave the Type option as Normal and the Default a1 as 1. This uses the algebraic sum of the results for the individual load cases in the load combination. The individual results will have a multiplication factor of 1.
2. Select the load cases for the load combination:
   a. Select 1: DEAD LOAD from Available Load List and then click [>].
   b. Select 2: EXTERNAL PRESSURE LOAD from Available Load List and then click [>].

   Load cases 1 and 2 will appear in the Load Combination Definition list.
   c. Click Add.

   Case 101 is created.

3. To define load case 5 as a load combination, repeat step 1 but type 102 in the Load No: field and type Case 1 + Case 3 in Title field.

4. Repeat step 2 except for selecting load cases 1 and 3 instead of cases 1 and 2.

   **Tip:** The load cases and combination methods used for a load combination can be changed by selecting a load combination and then clicking **Edit**.

   Load 102 is created.

5. Click Close.

   **Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing <CTRL + S>.
T.3 Specifying the analysis type

A linear, static analysis is required for this model. You can also instruct STAAD.Pro to provide a static equilibrium report.

The STAAD input file commands generated are:

```
PERFORM ANALYSIS PRINT STATICS CHECK
```

1. On the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group.

The Analysis/Print Commands dialog opens.

Tip: If the Analysis/Print Commands dialog does not open automatically, then click Define Commands in the Analysis - Whole Structure dialog.

2. In the Analysis/Print Commands dialog, select the Perform Analysis tab.
3. Select the **Statics Check** print option.

   **Note:** In response to this option, a report consisting of the summary of applied loading and summary of support reactions, for each load case, will be produced in the STAAD output file.

4. Click **Add**
5. Click **Close**.

   The **Analysis** command is added to the **Analysis - Whole Structure** dialog.
Tip: Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing **<CTRL+S>**.

**T.3 Specifying post-analysis print commands**

Two types of element results can be requested:

a. element stresses at the centroid or any point on the element surface which consist of stresses and moments per unit width, as explained in sections 1.6.1 and 3.41 of the STAAD Technical Reference Manual.

b. the element forces at the nodes which consist of the 3 forces and 3 moments at each node of the elements in the global axis system (see section 3.41 for details).

You will add both types of these results for this tutorial.

You will also need to set the units in which these results are printed to KN and Meter for element stresses and Kg and Meter for element forces.

The STAAD input file commands generated are:

```
UNIT METER KN
PRINT ELEMENT STRESSES LIST 3
UNIT KG METER
PRINT ELEMENT FORCE LIST 6
```

These results will be written in the STAAD output file and can be viewed using the procedure explained in “T.3 Viewing the output file” (on page 591).”

1. Set the length and force units to **Meter** and **Kilonewton** respectively.
   
   Refer to T.3 Changing the input units of length (on page 566) for additional information on this procedure.

2. On the Analysis and Design ribbon tab, select the Post-Analysis Commands tool in the Analysis Data group.

   The Post Analysis Print - Whole Structure dialog opens.

3. On the Post Analysis Print - Whole Structure dialog, click Define Commands.

   The Analysis/Print Commands dialog opens.
4. Select the **Element Forces/Stress** tab.
5. Select the **Print Element Stresses** option.
6. Click **Add** and then click **Close**.
7. Set the length and force units to **Meter** and **Kilogram** respectively.
8. Repeat steps 3 through 6 except select the **Print Element Forces** option in step 5.
Figure 88: The dialog with unassigned commands
9. Associate the print element stresses with element 3:
   a. Select the PRINT ELEMENT STRESSES command in the Post Analysis Print - Whole Structure dialog commands list.
   b. On the Geometry ribbon tab, select the Plates Cursor tool in the Selection group.
   c. In the view window, select element no. 3.
      When you select the plate, the Assignment Method automatically becomes Assign to Selected Plates.
   d. Click Assign.

10. To associate the PRINT ELEMENT FORCE command with element 6, use a similar procedure as step 9 except for selecting element no. 6 in the place of element no. 3.

   Tip: Remember to save your work by either click Save on the File ribbon tab, the Save tool, or pressing <CTRL+S>.

T.3 Viewing the input command file

You can inspect the text data file created during this tutorial.

1. On the Utilities ribbon tab, select the Command File tool in the Edit group.

   The STAAD.Pro Editor opens with the contents of the input file.
2. (Optional) You modify the data of the structure in this editor if necessary.
3. Select **File > Exit Editor** in the STAAD.Pro Editor window to close.

As stated in **Methods of creating the model** (on page 536), you could also have created the same model by typing the relevant STAAD commands into a text file using the STAAD.Pro Editor. If you would like to understand that method, proceed to the next section. If you want to skip that part, proceed to **Performing the analysis and design** (on page 590) where you will perform the analysis and design on this model.

### T.3 Creating the model using the command file

As an alternatively to the procedures described in the proceeding tutorial, you can also create the same STAAD input file using the STAAD.Pro Editor.

**Note:** A STAAD input file is a plain text file that uses the `.std` file extension. Therefore any standard text editor such as Notepad can also be used to create the command file. However, the STAAD.Pro Editor offers the advantage of syntax checking as you type the commands. The STAAD command syntax are highlighted by command, keyword, value, etc.

For this tutorial, delete all the command lines displayed in the editor window and type the lines shown below. While not necessary, this will allow you to learn more about the required and optional command lines for an input file.

STAAD commands are **not** case sensitive (i.e., they may be typed in upper or lower case letters). By convention, this and most input files use all caps, though.
For most all commands and keywords, the first three letters of a keyword are all that are needed. The rest of the letters of the word are not required, but are useful to present a user-friendly command language in mostly plain English for later reference. By convention, the required letters in a command or keyword are underlined here ("PLANE" = "PLA" = "plane" = "pla").

**STAAD SPACE SLAB SUPPORTED ALONG 2 EDGES**

Every input has to start with the word STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

**UNIT METER KN**

Specifies the unit to be used for data to follow.

**JOINT COORDINATES**

1 0 0 0 ; 2 2 0 0 ; 3 2 0 2 ; 4 0 0 2
5 4 0 0 ; 6 4 0 2 ; 7 6 0 0 ; 8 6 0 2
9 2 0 4 ; 10 0 0 4 ; 11 4 0 4 ; 12 6 0 4

Joint number followed by X, Y and Z coordinates are provided above. Semicolon signs (;) are used as line separators. That enables you to provide multiple sets of data on one line. For example, node 6 has (X, Y, Z) coordinates of (4, 0, 2).

**ELEMENT INCIDENCES SHELL**

1 1 2 3 4 ; 2 2 5 6 3 ; 3 5 7 8 6 ; 4 4 3 9 10 ;
5 3 6 11 9 ; 6 6 8 12 11

The incidences of elements are defined above. For example, element 3 is defined as connected between the nodes 5, 7, 8 and 6.

**UNIT CM KN**

**ELEMENT PROPERTY**

1 to 6 THICKNESS 30

The length unit is changed from meter to centimeter. Element properties are then provided by specifying that the elements are 30 cm thick.

**UNIT KN METER**

**CONSTANTS**

E 2.17185e+007 ALL
POISSON 0.17 ALL
DENSITY 23.5616 ALL
ALPHA 1e-005 ALL

Material constants, which are E (modulus of elasticity), Density, Poisson's Ratio and Alpha, are specified following the command CONSTANTS. Prior to this, the input units are changed to Meter and KN.

**SUPPORTS**

1 2 4 5 7 10 FIXED

Joints 1, 2, 4, 5, 7 and 10 are defined as fixed supported. This will cause all 6 degrees of freedom at these nodes to be restrained.

**UNIT KG**

**LOAD 1 DEAD LOAD**

Force units are changed from KN to KG to facilitate the input of loads. Load case 1 is then initiated along with an accompanying title.

**SELFWEIGHT Y -1**
Load case 1 consists of selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

**LOAD 2 EXTERNAL PRESSURE LOAD**

Load case 2 is initiated along with an accompanying title.

**ELEMENT LOAD**

1 TO 6 PR GY -300

Load 2 is a pressure load on the elements. A uniform pressure of 300Kg/m2 is applied on all the elements. GY indicates that the load is in the global Y direction. The negative sign (-300) indicates that the load acts opposite to the positive direction of global Y.

**LOAD 3 TEMPERATURE LOAD**

Load case 3 is initiated along with an accompanying title.

**TEMPERATURE LOAD**

1 TO 6 TEMP 40 30

Load 3 is a temperature load. All the 6 elements are subjected to a in-plane temperature increase of 40 degrees and a temperature variation across the thickness of 30 degrees. This increase is in the same temperature units as the Alpha value specified earlier under CONSTANTS.

**LOAD COMB 101 CASE 1 + CASE 2**

1 1.0 2 1.0

Load combination 101 is initiated along with an accompanying title. Load cases 1 and 2 are individually factored by a value of 1.0, and the factored values are combined algebraically.

**LOAD COMB 102 CASE 1 + CASE 3**

1 1.0 3 1.0

Load combination 102 is initiated along with an accompanying title. Load cases 1 and 3 are individually factored by a value of 1.0, and the factored values are combined algebraically.

**PERFORM ANALYSIS PRINT STATICS CHECK**

The above command instructs the program to proceed with the analysis. A static equilibrium report is also requested with the help of the words PRINT STATICS CHECK.

**UNIT METER KN**

PRINT ELEMENT STRESS LIST 3

The stresses and unit width moments are requested at the centroid of element 3 in KN and Meter units.

**UNIT KG METER**

PRINT ELEMENT FORCE LIST 6

The forces and moments for all 6 d.o.f at the corner nodes of element 6 are requested in KG and Meter units.

**FINISH**

This command terminates the STAAD run.

Save the input file and close the editor. The model is opened in the STAAD.Pro interface.
T.3 Performing the analysis and design

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab, the **Save** tool, or pressing **<CTRL>+<S>**.

1. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.

As the analysis progresses, several messages appear on the screen as shown in the figure below.

2. Select the **View Output File** option once the analysis and design are complete.

The three options are indicative of what will happen after you click **Done**.

- **View Output File**: This option opens the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands specified during the analysis.
- **Go to Post Processing Mode**: This mode allows for further examination and analysis of the results.
- **Stay in Modeling Mode**: This option keeps the user in the modeling phase for additional design modifications if needed.
the model generation process. It also provides you with important messages of any errors were encountered, and if so, whether the analysis and design was successfully completed or not. See T.1 Viewing the output file (on page 433) for details on viewing and understanding the contents of the output file.

**Go to Post Processing Mode**

This option opens the graphical Post-processor mode, which can be used to extensively review and verify the results. This mode allows you to view the results graphically, plot result diagrams, produce reports, etc. See T.1 Post-Processing (on page 441) for details on the Post processing mode.

**Stay in Modeling Mode**

This option closes the dialog and remains in the Model generation mode of the program, where you initiated the analysis. This is useful if you want to make further changes to the input file.

3. Click **Done**.
   The STAAD.Pro Output Viewer window opens.

### T.3 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly.

For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

**Note:**

If you did not select to open the output file after running the analysis in the previous procedure, you can open it any time through the user interface. On the **Utilities** ribbon tab, select the **Analysis Output** tool in the **View** group.

**Tip:** By default, the output file contains a listing of the entire input also. You may choose not to print the echo of the input commands in the Output file. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands** > **Set Echo** tool option from the menu bar and the select the **Echo Off** option in the **Set Echo** dialog.

It is **strongly recommended** that you review the entire output file to ensure that the results are reasonable and that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the post-processing mode. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.
Tutorials

T.3 - Analysis of a slab

* Licensed to: Bentley Systems Inc *
***************************************************************
1. STAAD SPACE
INPUT FILE: Plates Tutorial.STD
2. START JOB INFORMATION
3. ENGINEER DATE 11-APR-16
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER KN
7. JOINT COORDINATES
8. 1 0 0 0; 2 6 0 0; 3 6 0 4; 4 0 0 4; 5 2 0 0; 6 2 0 2; 7 0 0 2; 8 4 0 0
9. 9 4 0 2; 10 6 0 2; 11 2 0 4; 12 4 0 4
10. ELEMENT INCIDENCES SHELL
11. 1 1 5 6 7; 2 5 8 9 6; 3 8 2 10 9; 4 7 6 11 4; 5 6 9 12 11; 6 9 10 3 12
12. UNIT CM KN
13. ELEMENT PROPERTY
14. 1 TO 6 THICKNESS 30
15. DEFINE MATERIAL START
16. ISOTROPIC CONCRETE
17. E 2171.85
18. POISSON 0.17
19. DENSITY 2.35616E-005
20. ALPHA 1E-005
21. DAMP 0.05
22. TYPE CONCRETE
23. STRENGTH FCU 2.7579
24. END DEFINE MATERIAL
25. CONSTANTS
26. MATERIAL CONCRETE ALL
27. SUPPORTS
28. 1 2 4 5 7 8 FIXED
29. UNIT METER KG
30. LOAD 1 LOADTYPE NONE TITLE DEAD LOAD
31. SELFWEIGHT Y -1 LIST ALL
32. LOAD 2 LOADTYPE NONE TITLE EXTERNAL PRESSURE LOAD
33. ELEMENT LOAD
34. 1 TO 6 PR GY -300
35. LOAD 3 LOADTYPE NONE TITLE TEMPERATURE LOAD
36. TEMPERATURE LOAD
37. 1 TO 6 TEMP 40 30
38. LOAD COMB 101 CASE 1 + CASE 2
39. 1 1.0 2 1.0
40. LOAD COMB 102 CASE 1 + CASE 3
41. 1 1.0 3 1.0
42. PERFORM ANALYSIS PRINT STATICS CHECK

PROBLEM STATISTICS
****************************************
NUMBER OF JOINTS 12 NUMBER OF MEMBERS 0
NUMBER OF PLATES 6 NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0 NUMBER OF SUPPORTS 6
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 3, TOTAL DEGREES OF FREEDOM = 36
TOTAL LOAD COMBINATION CASES = 2 SO FAR.

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 1
LOADTYPE NONE TITLE DEAD LOAD

STAAD Space -- PAGE NO. 2

STAAD Pro 592 User Manual
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.299999999E+01
Y = 0.000000000E+00
Z = 0.199999999E+01

TOTAL APPLIED LOAD 1
***TOTAL APPLIED LOAD ( KG METE ) SUMMARY (LOADING 1 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -17298.83
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 34597.66 MY= 0.00 MZ= -51896.48

TOTAL REACTION LOAD 1
***TOTAL REACTION LOAD( KG METE ) SUMMARY (LOADING 1 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 17298.83
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -34597.66 MY= 0.00 MZ= 51896.48

MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING 1)
MAXIMUMS AT NODE
X = 0.00000E+00 0
Y = -3.20664E-01 3
Z = 0.00000E+00 0
RX= 9.80376E-04 3
RY= 0.00000E+00 0
RZ= -6.49326E-04 11

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 2
LOADTYPE NONE TITLE EXTERNAL PRESSURE LOAD
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.299999999E+01
Y = 0.000000000E+00
Z = 0.199999999E+01

TOTAL APPLIED LOAD 2
***TOTAL APPLIED LOAD ( KG METE ) SUMMARY (LOADING 2 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -7200.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 14400.00 MY= 0.00 MZ= -21600.00

TOTAL REACTION LOAD 2
***TOTAL REACTION LOAD( KG METE ) SUMMARY (LOADING 2 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 7200.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -14400.00 MY= 0.00 MZ= 21600.00

MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING 2)
MAXIMUMS AT NODE
X = 0.00000E+00 0
Y = -1.33465E-01 3
Z = 0.00000E+00 0
RX= 4.08045E-04 3
RY= 0.00000E+00 0
RZ= -2.70258E-04 11

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 3
LOADTYPE NONE  TITLE TEMPERATURE LOAD
TOTAL APPLIED LOAD     3
***TOTAL APPLIED LOAD ( KG   METE ) SUMMARY (LOADING     3 )
  SUMMATION FORCE-X =  0.0000000E+00
  SUMMATION FORCE-Y =  5.3673194E-28
  SUMMATION FORCE-Z =  0.0000000E+00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX= -9.5166050E-12  MY= -3.3531239E-10  MZ= -1.3304672E-13
TOTAL REACTION LOAD    3
***TOTAL REACTION LOAD( KG   METE ) SUMMARY (LOADING     3 )
  SUMMATION FORCE-X =  1.0313497E-10
  SUMMATION FORCE-Y =  4.9955999E-11
  SUMMATION FORCE-Z = -2.5783741E-10
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX= -9.0377167E-11  MY=  2.3891008E-09  MZ=  1.5455805E-10
STAAD SPACE                                              -- PAGE NO.    5
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING      3)
  MAXIMUMS    AT NODE
  X =  2.01178E-01       3
  Y =  8.97376E-01       3
  Z =  1.66238E-01       12
  RX= -3.51267E-03       3
  RY= -2.41811E-04       12
  RZ=  2.62397E-03       3
************** END OF DATA FROM INTERNAL STORAGE **************
43. UNIT METER KN
44. PRINT ELEMENT STRESSES LIST 3
ELEMENT STRESSES LIST     3
STAAD SPACE                                              -- PAGE NO.    6
ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
----------------
STRESS = FORCE/UNIT WIDTH/THICK,  MOMENT = FORCE-LENGTH/UNIT WIDTH

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT</td>
<td>VONB</td>
<td>SX</td>
<td>SY</td>
<td>SXY</td>
<td></td>
</tr>
<tr>
<td>TRESCAT</td>
<td>TRECAB</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-18.13</td>
<td>72.86</td>
<td>-3.96</td>
<td>-20.42</td>
</tr>
<tr>
<td>1308.65</td>
<td>1308.65</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>1404.84</td>
<td>1404.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>-220.35</td>
<td>SMIN=</td>
<td>-1404.84 TMAX=</td>
<td>592.24 ANGLE=</td>
<td>-11.1</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>1404.84</td>
<td>SMIN=</td>
<td>220.35 TMAX=</td>
<td>592.24 ANGLE=</td>
<td>78.9</td>
</tr>
<tr>
<td>2</td>
<td>-7.54</td>
<td>30.33</td>
<td>-1.65</td>
<td>-8.50</td>
<td>-1.39</td>
</tr>
<tr>
<td>544.68</td>
<td>544.68</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>584.71</td>
<td>584.71</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>-91.71</td>
<td>SMIN=</td>
<td>-584.71 TMAX=</td>
<td>246.50 ANGLE=</td>
<td>-11.1</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>584.71</td>
<td>SMIN=</td>
<td>91.71 TMAX=</td>
<td>246.50 ANGLE=</td>
<td>78.9</td>
</tr>
<tr>
<td>3</td>
<td>96.73</td>
<td>-59.42</td>
<td>-30.45</td>
<td>-14.83</td>
<td>18.43</td>
</tr>
<tr>
<td>10779.75</td>
<td>5300.98</td>
<td>-5044.91</td>
<td>2309.55</td>
<td>3890.86</td>
<td></td>
</tr>
<tr>
<td>3890.86</td>
<td>10912.07</td>
<td>5585.64</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>-312.06</td>
<td>SMIN=</td>
<td>-1989.55 TMAX=</td>
<td>838.74 ANGLE=</td>
<td>-11.1</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>1989.55</td>
<td>SMIN=</td>
<td>312.06 TMAX=</td>
<td>838.74 ANGLE=</td>
<td>78.9</td>
</tr>
<tr>
<td>102</td>
<td>78.61</td>
<td>13.44</td>
<td>-34.41</td>
<td>-35.24</td>
<td>15.08</td>
</tr>
<tr>
<td>1853.33</td>
<td>584.71</td>
<td>1853.33</td>
<td>5585.64</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10643.05</td>
<td>5713.20</td>
<td>-5044.91</td>
<td>2309.55</td>
<td>3890.86</td>
<td></td>
</tr>
<tr>
<td>11074.60</td>
<td>6408.66</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Tutorials
T.3 - Analysis of a slab

STAAD.Pro  594  User Manual
### T3 - Analysis of a slab

#### Maximum Stresses Among Selected Plates and Cases

<table>
<thead>
<tr>
<th>Plate No.</th>
<th>Case No.</th>
<th>Smax (Top)</th>
<th>Smmin (Top)</th>
<th>Tmax (Top)</th>
<th>Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>3</td>
<td>-923.25</td>
<td>-11074.60</td>
<td>5075.67</td>
<td>52.7</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>1848.79</td>
<td>-4559.87</td>
<td>3204.33</td>
<td>57.9</td>
</tr>
</tbody>
</table>

#### Global Corner Forces

<table>
<thead>
<tr>
<th>Joint</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
<th>MY</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELE.NO. 6 FOR LOAD CASE 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>0.0000E+00</td>
<td>4.5324E+02</td>
<td>0.0000E+00</td>
<td>-1.1313E+03</td>
<td>0.0000E+00</td>
<td>7.9082E+02</td>
</tr>
<tr>
<td>10</td>
<td>0.0000E+00</td>
<td>5.0615E+02</td>
<td>0.0000E+00</td>
<td>-3.2050E+02</td>
<td>0.0000E+00</td>
<td>2.3979E+02</td>
</tr>
<tr>
<td>3</td>
<td>0.0000E+00</td>
<td>-7.2078E+02</td>
<td>0.0000E+00</td>
<td>3.3973E-12</td>
<td>0.0000E+00</td>
<td>-3.2745E-13</td>
</tr>
<tr>
<td>12</td>
<td>0.0000E+00</td>
<td>-2.3860E+02</td>
<td>0.0000E+00</td>
<td>-4.6695E+02</td>
<td>0.0000E+00</td>
<td>-6.0134E+02</td>
</tr>
</tbody>
</table>

| ELE.NO. 6 FOR LOAD CASE 2 |
| 9    | 0.0000E+00 | 1.8864E+02 | 0.0000E+00 | -4.7087E+02 | 0.0000E+00 | 3.2915E+02 |
| 10   | 0.0000E+00 | 2.1067E+02 | 0.0000E+00 | -1.3340E+02 | 0.0000E+00 | 9.9804E+01 |
| 3    | 0.0000E+00 | -3.0000E+00 | 0.0000E+00 | 8.1863E-14  | 0.0000E+00 | 1.6373E-13 |
| 12   | 0.0000E+00 | -9.9310E+01 | 0.0000E+00 | -1.9435E+02 | 0.0000E+00 | -2.5029E+02 |

| ELE.NO. 6 FOR LOAD CASE 3 |
| 9    | -2.9880E+05 | 6.6191E+02 | -3.0717E+05 | 6.3684E+03 | 2.7909E+03 | -5.5444E+03 |
| 10   | 3.0633E+05  | -9.9012E+02 | -3.2773E+05 | 4.3051E+03 | -3.7448E+03 | 4.3520E+03 |
| 12   | -3.2773E+05 | 3.2821E+02  | -3.1470E+05 | -4.0134E+03 | 9.5385E+02  | -2.8310E+03 |

| ELE.NO. 6 FOR LOAD CASE 101 |
| 9    | 0.0000E+00 | 6.4188E+02 | 0.0000E+00 | -1.6022E+03 | 0.0000E+00 | 1.1200E+03 |
| 10   | 0.0000E+00 | 7.1682E+02 | 0.0000E+00 | -4.5390E+02 | 0.0000E+00 | 3.3959E+02 |
| 3    | 0.0000E+00 | -1.8208E+03 | 0.0000E+00 | 2.3804E-03  | 0.0000E+00 | -2.9084E-03 |
| 12   | 0.0000E+00 | -3.3792E+02 | 0.0000E+00 | -6.6131E+02 | 0.0000E+00 | -8.5163E+02 |

| ELE.NO. 6 FOR LOAD CASE 102 |
| 9    | -2.9880E+05 | 1.1151E+03 | -3.0717E+05 | 5.2371E+03 | 2.7909E+03 | -4.7536E+03 |
| 10   | 3.0633E+05  | -7.2079E+02 | -3.2773E+05 | 3.9846E+03 | -3.7448E+03 | 4.5918E+03 |
| 3    | 3.2019E+05  | 8.9607E+01  | 3.1470E+05  | -4.4804E+03 | 9.5385E+02  | -3.4324E+03 |

#### Additional Notes
- Details about additional assistance from Bentley and Partners can be found at program menu
- Help->Technical Support
- Copyright (c) 1997-2017 Bentley Systems, Inc.
T.3 Post-Processing

If there are no errors in the input, the analysis is successfully completed. The extensive facilities of the Post-processing mode can then be used to:

- view the results graphically and numerically
- assess the suitability of the structure from the standpoint of safety, serviceability and efficiency
- create customized reports and plots

The procedure for entering the post processing mode is explained in tutorial 2 (on page 506).

Node results such as displacements and support reactions are available for all models. The methods explained in the first two tutorials may be used to explore these. For this example, you will examine the support reactions.

If beams are present in the model, beam results will be available too. As no beams are present in this model, this is not covered for this tutorial.

For plates, the results available are stresses, and “unit width” moments. There are several different methods for viewing these results, as explained in the next few sections.

T.3 Viewing stress values in a tabular form

1. On the View ribbon tab, select the Tables tool in the Windows group.

   The Tables dialog opens.

2. Select Plate Center Stress and click OK.
The **Plate Center Stress** table opens.

![Image of the Plate Center Stress table](image)

**Figure 90:** The table has the following tabs:

- **Shear, Membrane and Bending**
  - These terms are explained in [G.5.1 Plate and Shell Elements](#) (on page 2308). The individual values for each plate for each selected load case are displayed.

- **Summary**
  - This tab contains the maximum for each of the 8 values listed in the Shear, Membrane and Bending tab.

- **Principal and Von Mises**
  - These terms too are explained in [G.5.1 Plate and Shell Elements](#) (on page 2308). The individual values for each plate for each selected load case are displayed, for the top and bottom surfaces of the elements.

- **Summary**
  - This tab contains the maximum for each of the 8 values listed in the Principal and Von Mises tab.

- **Global Moments**
  - This tab provides the moments about the global X, Y and Z axes at the center of each element.

3. (Optional) Right-click in the table area and select **Print** from the pop-up menu.

### T.3 Changing the units of values in the output

The length and force units of the stresses and moments are displayed alongside the individual column headings for the terms.

1. On the **File** ribbon tab, select **Display Options** on the **Settings** tab.

   The **Options** dialog opens.
2. Select the **Force Units** tab and specify the required unit from the **Stress** and **Moment** fields.
3. (Optional) Click **Apply**
4. Click **OK**

### T.3 Limiting the load cases for which the results are displayed

Use the following procedure to change the load list used for present results in the Post Processing mode.

1. On the **Results** ribbon tab, select the **Select Load Case** tool in the **Configuration** group.

   ![Results Setup dialog](image)

   The **Results Setup** dialog opens.

2. Click `<<`.

   All load cases are moved from the Selected list to the Available list.

3. Select the load cases you want from the **Available** list and click `>`.

   The selected load cases are transferred from the **Available** list to the **Selected** list.
4. Click **OK**.

T.3 Stress Contours

Stress contours are a color-based plot of the variation of stress or moment across the surface of the slab or a selected portion of it.

1. On the **Results** ribbon tab, select the **Plate Stress** tool in the **View Results** group.

   ![Diagrams dialog](image)

   The **Diagrams** dialog opens to the **Plate Stress Contour** tab.

2. Select the load case number from the **Load Case** drop-down list.

   Stress values are known exactly only at the plate centroid locations. Everywhere else, they are calculated by linear interpolation between the center point stress values of adjacent plates.

   The **Enhanced** type contour chooses a larger number of points compared to the **Normal** type contour in determining the stress variation.

3. Select the specific type of stress for which you want the contour drawn from the **Stress type** drop-down list.

4. Setting the **View Stress Index** option will display a small table consisting of the numerical range of values from smallest to largest which are represented in the plot.

   Set the following:

   - Load case — 102: CASE 1 + CASE 3
   - Stress Type — Von Mis Top
Tutorials
T.3 - Analysis of a slab

Contour Type — Normal Fill
Index based on Center Stress checked
View Stress Index checked
Re-Index for new view checked

5. Click **Apply**.
The following diagram will be displayed.
T.3 Animating stress contours

The Diagrams dialog is used to provide a dynamic animation of stress plots.

In order to animate a diagram, such as the plate stress contour, you must first have a diagram displayed. Refer to T.3 Stress Contours (on page 599) for how to plot those results on the structure.

1. Either:
   - on the Results ribbon tab, select the Animation tool in the Animation group
   - or
   - if the Diagrams dialog is still open from the previous procedure, select the Animation tab

2. Select the Stress option.
3. (Optional) Set the **Target FPS 5**

   **Tip:** This will slow the animation down to make it easier to distinguish the frames.

4. Click **Apply**.

   The plate stress contours are animated loading and unloading.
5. To stop the animation, select the **No Animation** option and click **Apply** again.

### T.3 Creating AVI Files

You can save dynamic result, such as a deflection diagram in animation, to a video file. This feature is available in STAAD.Pro for node deflection, beam section displacement, mode shape and plate stress contour diagrams. These files can then be viewed using video player programs such as the Windows Media Player.

1. On the **Utilities** ribbon tab, select the **Create AVI File** tool in the **Utilities** group.

The **Create AVI File** dialog opens.
Total No. of Frames  Sets the number of frames used to capture the movement. In an animated view, the movement from one extremity to the other is captured as several frames.

Frame Rate /sec  sets the speed of the motion

The rest of the options in the above dialog are for the type of diagram from which the video file is to be created. Certain items such as Mode Shape and Plate Stress contour are disabled if the required data of that type are not present in the STAAD file, such as a modal extraction, or finite elements.

2. Select the Stress Contour option for Animation Type and then select Von Mis Top from the Plate Stress Type drop-down list.

3. Click OK.  
A dialog opens to specify the file name.

4. Type a file name and location for the video file and then click OK.

The Video Compression dialog opens.
5. (Optional) Select a Compressor option and set the Compression Quality value.

   **Note:** Video files can be quite large, and compression is a technique used reduce the size of these files, though some video smoothness is lost in this process.

6. Click **OK** to begin creating the video file.

   When the file has been generated, a message opens indicating that the operation was successful.

7. Click **OK** to dismiss.

   The file with the extension *AVI* is saved in the same folder where the STAAD input file is located.

---

**T.3 Viewing plate results using element query**

Element Query is a facility where several results for a specific element can be viewed at the same time from a single dialog.

1. On the **Results** tab, select the **Plate Cursor** tool in the **Selection** group.

2. Either:
   
   - Double-click on element 4
   
   or

   - select element 4 and then select the **Properties** tool in the **Model** group on the **Plate Tools** ribbon tab.

   **Tip:** Press `<Shift+P>` to display the plate numbers.

   The **Plate** dialog opens.
The various tabs of the query box enable one to view various types of information such as the plate geometry, property constants, stresses, etc., for various load cases, as well as print those values.

Some example tabs of this dialog are shown in the following figures.
Tutorials
T.3 - Analysis of a slab

The image shows a software interface for the analysis of a slab using STAAD.Pro. The interface includes options for specifying material properties and physical properties, as well as load lists. The interface displays the results for principal stresses, von mises stresses, and Tresca stresses for the slab.

Material Properties:
- Elasticity: 21,718,456 kN/mm²
- Density: 2402,6156 kg/m³
- Poisson: 0.17
- Alpha: 1e-05

Physical Properties:
- Node 7: Thickness = 0.300000011
- Node 6: Thickness = 0.300000011
- Node 11: Thickness = 0.300000011
- Node 4: Thickness = 0.300000011

Load List:
- 1: DEAD LOAD

Stress Results:
- Principal Stresses:
  - Top (N/mm²): -0.972646, 0.946139, 0.972646
  - Bottom (N/mm²): 0.972646, 0.946139, 0.972646

- Von Mises Stresses:
  - Top (N/mm²): 0.946139, 0.946139, 0.946139
  - Bottom (N/mm²): 0.946139, 0.946139, 0.946139

- Tresca Stresses:
  - Top (N/mm²): 0.972646, 0.972646, 0.972646
  - Bottom (N/mm²): 0.972646, 0.972646, 0.972646
T.3 Producing a report

Produce a report consisting of the plate principal stresses, for all plates, sorted in the order from Low to High of the Principal Maximum Stress (SMAX) for load cases 101 and 102.

Occasionally, you will need to obtain results conforming to certain restrictions, such as, say, the resultant node displacements for a few selected nodes, for a few selected load cases, sorted in the order from low to high, with the values reported in a tabular form.

1. On the Results tab, select the Plate Cursor tool in the Selection group.

2. Select the all the plate elements.
   
   **Tip:** Drag a window around all the elements in the view area or press <Ctrl+A>.

3. On the Results ribbon tab, select the Reports > Principal Stresses tool in the Reports group.

   The Plate Forces dialog opens.

   ![Plate Forces Dialog](image)

4. Specify the report contents:
   a. Select the Sorting tab.
   b. Select the SMAX option in the Sort By Plate Stress group.
c. Select **List from Low to High** option in the **Set Sorting Order** group.

d. Clear the **Absolute Values** check box in the **If Sorting done** group.

5. Select the load cases to use:
   
a. Select the **Loading** tab.

   ![Plate Forces window](image)

   b. Select load cases **101** and **102** in the **Available** list and click [>] to add them to the **Selected** list.

6. (Optional) You can save this report for future use,
   
a. Select the **Report** tab

   b. Type a **Title** for the report.

   c. Set the **Save Report** check box.

7. Click **OK**.

   The Plate Forces table opens to display the table of maximum principal stress with SMAX values sorted from Low to High.
8. (Optional) To print this table, right-click anywhere within the table and select **Print**

To transfer the contents of this table to a Microsoft Excel file, click the **Plate** label in the top-left corner of the table. The entire table is selected. Right-click and select **Copy** from the pop-up menu. You can now paste the table contents into a Microsoft Office Excel spreadsheet or other spreadsheet file.

**T.3 Viewing Support Reactions**

Use the nodal results page to obtain support reaction results.

1. Select the **Reactions** page in the **Postprocessing** page control bar.

   The reactions at the supports will be displayed on the drawing and the Support Reactions table is displayed on the right side of the program window.
The six values — namely, the three forces along global X, Y and Z, and the three moments Mx, My and Mz, in the global axis system — are displayed in a box for each support node.

2. Display of one or more of the six terms of each support node may be toggled off in the following manner.
a. On the **Results** ribbon tab, select the **Annotate** tool in the **Configuration** group. The **Annotation** dialog opens.

![Annotation dialog](image)

b. Select the **Reactions** tab.

c. Clear the degree of freedom check boxes in the **Direct** or **Bending** groups you want to hide from display.

d. Click **Annotate** and then **Close**.

The drawing will now contain only the selected items.

3. **(Optional)** To change the load case for which the reactions are displayed, select the desired case from the **Load** drop-down in the program status bar.

![Load drop-down](image)

4. On the **Support Reactions** table, select the **Summary** tab.

**Tip:** You can also open this table by selecting **Tables > Reactions** tool in the **View Results** group on the **Results** ribbon tab.
The summary tab contains the maximum value for each of the 6 degrees of freedom along with the load case number responsible for it.

Refer to T.3 Changing the units of values in the output (on page 597) to change the units in which these values are displayed.

This brings us to the conclusion of this tutorial.
This section of the help describes methods used to model your structure in STAAD.Pro. This section begins by detailing how to model analytical elements using the Analytical Modeling workflow. Following that are sections on how to model the structure using other workflows and utilities in STAAD.Pro.

M. Navigating the Graphical View Window

This section describes how to control the graphical display of your model.

M. To select a center of rotation at a node

Used to select an node as the center of rotation. When toggled on, pressing <Ctrl+Shift> and clicking a node will set that node as the center of rotation.

1. Either:
   - On the View ribbon tab, select the Toggle View Rotation Mode tool in the Rotation group.
   - Or
   - On the View ribbon tab, select the Always Fit in Current Window tool in the Rotation group.

   **Note:** The icon for the Toggle View Rotation Mode is depressed when in this mode (and there is no check on the View > Always Fit in Current Window menu item).

   A message dialog appears to remind you how to next select a center of rotation.

2. Hold down the <Ctrl+Shift> keys and click on a node.

   A reticle is placed on this node to indicate it is the center of rotation.

3. Either:
   - Use the rotate tools found in the Rotate toolbar or the arrow keys to rotate the structure about this node.
   - Or
   - Hold down the right mouse button while dragging the pointer to free rotate the model about this node.

4. Repeat steps 3 and 4 to re-center and rotate the structure.

5. To return the center of rotation to the center of the structural model:
   - select the Toggle View Rotation Mode tool
   - Or
select the **Always Fit in Current Window** tool such that a check mark appears or

select the **Use Center of the structure as rotational Center** option in the **Orientation** dialog.

**Tip:** You may specify the exact center of rotation coordinates at any point using the **Orientation** dialog.

### M. To view a 3D rendering of your model

To display the model with full 3D sections and lighting, use the following procedure.

Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

**Note:** The 3D rendering is used to display the model only. You cannot select model objects or otherwise manipulate the model in this view.

1. On the **View** ribbon tab, select the **3D Rendering** tool in the **Windows** group.

A new view window opens with the model displayed as a 3D rendering.

2. Navigate the rendering view as follows:

<table>
<thead>
<tr>
<th>To…</th>
<th>Do this…</th>
</tr>
</thead>
<tbody>
<tr>
<td>rotate the structure</td>
<td>left-click in the view and drag the mouse in any direction</td>
</tr>
<tr>
<td>pan the structure</td>
<td>click-and-drag the middle mouse button (typically the scroll wheel)</td>
</tr>
<tr>
<td>zoom in/out</td>
<td>scroll the mouse wheel up or down</td>
</tr>
</tbody>
</table>

3. (Optional) To generate a picture for inclusion in reports:

   a. Right-click and select **Take Picture** from the pop-up menu.

   The **Picture #** dialog opens.

   b. Type a picture **ID** and **Caption**.

   c. Click **OK**.

4. Click the [X] in the top, right-hand corner of the window to close the view.

### M. 3D Rendering View Right-click View Menu

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>View Axes</td>
<td>Toggles the display of the global axes.</td>
</tr>
<tr>
<td>Perspective</td>
<td>Toggles the perspective lens for the rendered window.</td>
</tr>
<tr>
<td>Wireframe View</td>
<td>When selected, solid rendering is turned off. Only the outline of shapes are shown.</td>
</tr>
</tbody>
</table>
### Menu item

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enable Lighting</td>
<td>Toggles the light source. Turning this feature off removes any reflection.</td>
</tr>
<tr>
<td>Lights (list)</td>
<td>Select the light color from the list.</td>
</tr>
<tr>
<td>Change Light Color</td>
<td>The lighting is by default gray to resemble the color of steel. This can be changed to any other color.</td>
</tr>
<tr>
<td>Enable Texture Mapping</td>
<td>The structure will be displayed with a unique texture.</td>
</tr>
<tr>
<td>Change Entity Color...</td>
<td>Opens the Change Entity Color dialog, which is used change the color of structure entities by class.</td>
</tr>
<tr>
<td>Change Background Color...</td>
<td>Opens a color selector dialog, so the default background color can be changed.</td>
</tr>
<tr>
<td>Change Font...</td>
<td>Opens a font selection dialog, so the text display can be changed.</td>
</tr>
<tr>
<td>Model View Details</td>
<td>Opens the Structure Diagram Info dialog, which contains a summary of the entity counts in the structure.</td>
</tr>
<tr>
<td>Take Picture</td>
<td>Takes a STAAD image of the rendered view for inclusion in reports. A picture ID dialog opens to name the image.</td>
</tr>
</tbody>
</table>

### M. Labels

STAAD.Pro has numerous options for displaying labels in the view window.

**M. To switch on labels for nodes, beams, plates, etc.**

Labels are a way of identifying the entities we have drawn on the screen.

1. Either:

   Select the **Labels** tool

   or

   Right-click anywhere in the View area and select **Labels** from the pop-up menu. The **Diagrams** dialog opens.

2. Select the **Labels** tab and then select the options for the appropriate labels (examples shown in the following figure).
Modeling

M. Navigating the Graphical View Window

![Diagram of STAAD Pro's Graphical View Window with various options for modeling and visualization settings.](image)

For quick access to the labels using keyboard hotkeys, press Shift + the letter shown in brackets.
3. Click **OK** to update the View window and close the dialog.
4. To change the font of the node/beam labels, select **View > Options**.
   
a. Select the **File** ribbon tab.
   
The STAAD.Pro Backstage view opens.

b. On the **Settings** tab, select **Display Options**.

   The **Options** dialog opens.

c. Select the appropriate tab (Node Labels / Beam labels) and then click **Font** to make the desired changes.
d. Click OK to close the Font dialog.

e. Click OK to close the Options dialog.

M. To change the structural tool tip options

Structural tool tips offer a facility for displaying any customized input or output information about a node, beam, plate or solid element when the mouse cursor is placed over the structural entity. The tool tips are similar to the ones displayed when the mouse cursor hovers over a toolbar icon.

1. Select the File ribbon tab.
   The STAAD.Pro Backstage view opens.
2. On the Settings tab, select Structural Tooltip Options.
   The Tool Tip Options dialog opens.
3. Check the **Show Tool Tip** option.

4. (Optional) Change the **Tip Delay**

   This number is expressed in milliseconds (i.e., $1,000 = 1$ second).

5. For a selected model entity (Node, Beam, etc. in the left list), set the information and results you want to display in the tool tip in the right list.

   The options (items that can be displayed) for each entity are shown under the **Options** list. A check mark signifies that the particular data item will be displayed in the tool tip. An option with a "+" next to it signifies that further options can be enabled or disabled. A red "X" indicates the data will not be shown in the tool tip. Simply click on the check box to turn an option on or off.

6. Click **OK**.

---

**Example**

Structural tool tips can be configured to display a wide variety of information by hovering over a model entity, as shown in the following figure.
Note: The tool tips automatically display the results for the active load case. All values are reported in the current display units.

M. To switch individual object label display

To control the display of individual node, beam, plate, or solid labels, use the following procedure.

These tools activate the Partial Labeling Mode, which allows you to control the display of individual object labels in STAAD.Pro.

1. On the View ribbon tab, select one of the following tools in the Labels group:

   **Individual Node Label**

   or

   **Individual Beam Label**
2. Select the model object that you want to switch the label state.
   The model object must be of the same type as the selected tool.
   If this is the first time you are switching individual labels, you will be prompted that the program will turn on
   Partial Labeling Mode. Click **Yes** to proceed.

M. Views

You can navigate the view, control the display of model objects, and save named views in STAAD.Pro.

M. Displaying a Portion of the Model

Sometimes, the large number of entities that are drawn on the screen may make it difficult to clearly see the
details at any particular region of the structure. In such cases, one is confronted with the task of decluttering the
screen or looking at specific regions or entities while removing the rest of the structure from the view.

There are different methods in STAAD.Pro by which you can view a portion of the structure.

M. To cut a section of a model

The file US 8. Concrete Design for a Space Frame.STD is used to demonstrate this feature.

1. (Optional) Press `<Shift+N>` to display the node numbers.
   This will aide in specifying a node number in the following step.
2. Use the cut section tool:
   a. On the Utilities ribbon tab, select the Structure Tools tool in the Geometry Tools group.
   
   ![Structure Tools icon]

   b. Select the Cut Section tool from the drop-down list.
   
   ![Cut Section icon]

   The Section dialog opens.
3. Specify a cut plane by joint and plane orientation (range by joint method):
   a. Select the Range By Joint tab.
   b. Select the X - Z Plane option.
   c. Select 10 from the With Node # drop-down list (which contains all node numbers in the model).
      
      Tip: You can select any node with in this same plane to achieve the same result.
   d. Click OK.

4. To restore the original view, select the Whole Structure tool in the Tools group on the View ribbon tab.

   Tip: Alternatively, you can click Show All on the Section dialog.

5. Specify a cut plane by minimum and maximum distances and plane orientation (the range by min/max method):
   a. Repeat step 2 to open the Section dialog.
   b. Select the Range By Min/Max tab.
c. Select the **X-Z Plane** option.

d. Type 10 in the **Minimum** field and type 14 in the **Maximum** field.

The **Minimum** and **Maximum** values are the boundary distances along the axis perpendicular to the sectional plane. Every object lying entirely between these two distances will be displayed.

**Note:** When using this method, make sure that the current input units of length are in the intended units.

6. Repeat step 4 to restore original view.
7. Display only the nodes for quick selection (the Select to View method):
   a. Select the **Select To View** tab.

   ![Select To View](image)

   **Figure 93:**

   b. Select the **Select To View** option.
   c. Check the **Node** option.
   d. Click **OK**.
8. Select the **Whole Structure** tool to restore original view.

**Note:** You can save these views by selecting the **View Management > Save View** tool in the **Views** group on the **View** ribbon tab. Provide a title for the new view. These saved views may later be opened by selecting the **Open View** tool in the **Views** group on the **View** ribbon tab.

### M. To create a new view

The file **US.8. Concrete Design for a Space Frame.STD** is used to demonstrate this feature.

New view windows are helpful for performing such operations as adding and deleting members, assigning properties, loads, supports, and more. A new view of a selected portion offers the advantage of de-cluttering the screen and limiting the displayed objects to just a few chosen entities.

1. On the **View** ribbon tab, select the **Front View** tool in the **Tools** group.

2. In the view window, drag a window area around the middle level of beams to select them.

**Tip:** Make sure that the **Beam Cursor** tool is the active selection tool on the **Select** ribbon tab in the **Cursors** group.

The selected members are highlighted.
3. (Optional) On the View ribbon tab, select the Isometric View in the Tools group.

4. Either:
   On the View ribbon tab, select the New View tool in the Views group.
   
or
   Right-click anywhere in the current view window and select New View from the pop-up menu.
The **New View** dialog opens.

The dialog includes options open the new view in a new (child) window or to replace the current (parent) view window.

5. Select the **Create a new window for the view** option and then click **OK**.

The portion of the structure that we selected is displayed in a new window.

**Tip:** Multiple child view windows may be created in this way.

6. Save the view for later use:
   a. On the **Views** ribbon tab, select the **View Management Save View** tool in the **Views** group.

   The **Save View As** dialog opens.
   b. Type a name for the view.
   c. Click **OK**.

   The view window is titled with the view name. You can

7. Close the new view window by clicking close [X] in the top-right corner.
8. Repeat steps 2 through 4 to start another New View.
9. Select the **Display the view in the active window** option and click **OK**.

The full structure is hidden and the selected portion is displayed in the View window.

10. To restore the original view, select the **Whole Structure** tool in the **Tools** group on the **View** ribbon tab.
M. To insert custom text in a view

To add custom text to the view window and, subsequently, pictures taken for reports, use the following procedure.

The text can serve as comments or titles to pictures and result diagrams. The added text can be plotted too.

1. Select the **Insert Text** tool in the **Display** group on the **Utilities** ribbon tab.

   The mouse pointer changes to the text cursor (\(\text{ insertion symbol}\)).

2. Click in the view window where you want to insert the text.

   The **Write Text to Insert** dialog opens.

3. (Optional) Click **Font** to set the type display.

4. Type the text into the text field and then click **OK**.

5. Either:

   Repeat steps 2 through 4 to insert additional text items.

   or

   Repeat step 1 to turn off the insert text tool.

6. (Optional) To modify previously inserted text:

   a. Select the **Text Cursor** tool in the Selection group on the **Geometry, Results**, or **Member Design** ribbon tabs.

   b. Double-click on the text you want to modify.

      An edit dialog opens with the existing text.

   c. Make changes and then click **OK**.

   d. Click-and-hold the left mouse button on the text to move it to another location in the view window.

7. (Optional) To delete previously inserted text:
a. Select the Text Cursor tool in the Selection group on the Geometry, Results, or Member Design ribbon tabs.

b. Click on the text you want to delete.

c. Press <Delete>.

M. To display loads graphically

You can display the load diagrams and load values on the model.

1. On the Loading ribbon tab, select the current Load from the drop-down list in the Display group.

   ![Load Selection](image)

   Tip: The current load case number is displayed in the lower, right-hand side of the view window as well as in the application status bar.

2. Either:

   Select the Labels tool

   or

   Right-click anywhere in the View area and select Labels from the pop-up menu.

   The Diagrams dialog opens.

3. Either:

   a. select the Loads and Results tab.

   b. Check the Loads option.

   c. (Optional) Click the color squares for Direct or Moment to change the load pattern colors.

   d. Click Apply.

   The loads are graphically displayed on the model.
4. a. Select the Labels tab.
   b. Set the Load Values check box in the Loading Display Options group.
   c. Click OK.

The load values are displayed for the current load case, as seen here for the file US.8. Concrete Design for a Space Frame.STD.
5. (Optional) To change the units displayed:
   a. Select the **File** ribbon tab.
      The STAAD.Pro Backstage view opens.
   b. On the **Settings** tab, select **Display Options**.
      The **Options** dialog opens.
   c. Select the **Force Units** tab.
      The **Options** dialog opens.
d. Select the **Force** and **Distributed Force** values as needed.

e. Click **OK**.
   
The values are updated with the new units.

### M. To identifying beam start and end

Beam ends can be colored to easily identify the start and end joints of a beam. By default, the start ($i$) joint is colored green and the end ($j$) joint is colored blue.

1. Either:
   
   Select the **Labels** tool

   or

   Right-click anywhere in the View area and select **Labels** from the pop-up menu.

   The **Diagrams** dialog opens.

2. Check the **Beam Ends** option in the **Beams** group.

3. Click **OK**

   **Tip:** Alternatively, you can press `<Shift+E>` to toggle the beam end colors.
The beam ends are colored for the entire structure.

The file US.1. Plane Frame with Steel Design.STD is used to demonstrate this feature.

Alternatively, you can hover your mouse over any beam momentarily and the beam ends for that member are displayed using the same colors.
M. Rotation tools

These tools found on the View ribbon tab can rotate your model to a predefined orientation.

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Isometric View</td>
<td>Displays the structure in the isometric view. The angle which defines isometric view is generally $X = 30, Y = 30, Z = 0$</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
<td>Example</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>---------</td>
</tr>
<tr>
<td>Front View</td>
<td>Displays the structure as seen from the positive Z axis. When the global Y axis is vertical, this is the elevation view, as looking towards the negative direction of the Z-axis.</td>
<td><img src="image1" alt="Front View Example" /></td>
</tr>
<tr>
<td>Back View</td>
<td>Displays the structure as seen from the negative Z axis. When the global Y axis is vertical, this is the elevation view, as seen looking towards the positive direction of the Z-axis.</td>
<td><img src="image2" alt="Back View Example" /></td>
</tr>
<tr>
<td>Left View</td>
<td>Displays the structure as seen from the negative X axis. When the global Y axis is vertical, this is the side elevation, as seen looking towards the positive direction of the X-axis.</td>
<td><img src="image3" alt="Left View Example" /></td>
</tr>
<tr>
<td>Right View</td>
<td>Displays the structure as seen from the positive X axis. When the global Y axis is vertical, this is the side elevation, as seen looking towards the negative direction of the X-axis.</td>
<td><img src="image4" alt="Right View Example" /></td>
</tr>
</tbody>
</table>
### M. To display master nodes

To highlight master nodes in a model that contains master/slave joints, use the following procedure.

**Tip:** Open example 26 to demonstrate this procedure for yourself.

1. Either:
   - on the **View** ribbon tab, select the **Label Settings** tool in the **Labels** group
   - or
   - in the view window, right-click and select **Labels** from the pop-up menu
   - The **Diagrams** dialog opens to the Labels tab.

2. Check the option from **Master Slave** in the General group.

3. Click **Apply**.

   **Tip:** Press `<Shift+L>` to highlight master nodes quickly.

   The master nodes are highlighted with green cubes.
4. Click OK.

Related Links
- Diagrams dialog (on page 2927)
- EX. US-26 Modeling a Rigid Diaphragm Using Master-Slave (on page 4574)
- EX. UK-26 Modeling a Rigid Diaphragm Using Master-Slave (on page 4855)

M. Creating Model Objects
This section describes the tools used in STAAD.Pro to model your structure.

M. Drawing Aids

M. To add a grid for drawing objects
To add a grid for drawing beams or plates, use the following procedure.
Only one grid may be displayed at a time. However, you may have multiple grids associated with your model.

**Note:** Grids added to the model will be available to use in both the Snap Node/Beam, Snap Node/Plate, and Snap Node/Solid dialogs.

1. Either:
   - On the Geometry ribbon tab, select the Grids > Beam Grid tool in the Structure group.

![Beam Grid](image1)

   or
   - On the Geometry ribbon tab, select the Grids > Triangular Plate Grid tool in the Structure group.

![Triangular Plate Grid](image2)

   or
   - On the Geometry ribbon tab, select the Grids > Quad Plate Grid tool in the Structure group.

![Quad Plate Grid](image3)

   or
   - On the Geometry ribbon tab, select the Grids > Solid Grid tool in the Structure group.
The corresponding Snap Node/Beam, Snap Node/Plate, or Snap Node/Solid dialog opens. If an existing grid is selected in this dialog, this grid is displayed in the view window.

2. Click Create.
   A pop-up dialog opens for defining grid details.

3. Select the type of grid from the drop-down list.

<table>
<thead>
<tr>
<th>Grid type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>a Cartesian grid with even spaces along both of the axis.</td>
</tr>
<tr>
<td>Radial</td>
<td>a polar grid with even spaces between rays and radial lines.</td>
</tr>
<tr>
<td>Irregular</td>
<td>a Cartesian grid with irregular spaces along both of the axis.</td>
</tr>
</tbody>
</table>

The pop-up dialog fields update with the grid type selection.

4. Type a grid Name to identify the grid in the Snap Node/Beam dialog.

5. Specify the location of the grid with respect to the global axes:
   a. Select the parallel global Plane for alignment (or reference when an angle is specified).
   b. (Optional) Select a global axis and then type an Angle of Plane to incline the grid with respect to that axis.
   c. (Optional) (For Irregular grids) Set the option to Use Arbitrary Plane and then type the vectors of the grid X and Y axis.
      This method can be used instead of specifying the Plane and Angle of Plane.
   d. Type global coordinates from the global origin to the Grid Origin.

   Tip: Click the node selection tool (pencil) and then click on an existing node in the model to use as the grid origin.

6. Specify the Construction Lines for the grid:

<table>
<thead>
<tr>
<th>Grid type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>For both grid X and Y, type the number of grids spaces to the Left and Right of the origin, the Spacing (in current units) between each grid line, and (optionally) a Skew angle.</td>
</tr>
<tr>
<td>Radial</td>
<td>Type the Start Angle and Sweep to specify the beginning and end of the radial lines along with the number of bays (spaces between radial lines). Type the inner-most and outer-most radius in the Radius 1 and Radius 2, respectively, along with the number of Bays (spaces between arcs).</td>
</tr>
<tr>
<td>Irregular</td>
<td>Type the distance between each consecutive grid line in the current units along each axis.</td>
</tr>
</tbody>
</table>

7. Click OK.
   The pop-up dialog closes and the new grid name is added to the list.

8. Check the name of the grid in the list that you want displayed in the view window.

Click Edit to make changes to the currently selected grid in the list. You can change the grid orientation and construction lines but the type of grid cannot be changed.

Related Links
- Snap Node-Beam dialog (on page 2895)
- Linear (on page 2897)
- Radial grid dialog (on page 2897)
- Irregular grid dialog (on page 2898)
M. To import a STAAD.Pro grid file

To import a STAAD.Pro grid file which was contains grids used in another model, use the following procedure.

Any time you save a model with custom grids, the grid data is saved into a STAAD.Pro grid file (file extension .grd) with the same file name and location as the STAAD input file.

1. Either:
   - On the Geometry ribbon tab, select the Grids > Beam Grid tool in the Structure group.

   ![Image of Beam Grid tool]

   or
   - On the Geometry ribbon tab, select the Grids > Triangular Plate Grid tool in the Structure group.

   ![Image of Triangular Plate Grid tool]

   or
   - On the Geometry ribbon tab, select the Grids > Quad Plate Grid tool in the Structure group.

   ![Image of Quad Plate Grid tool]

   or
   - On the Geometry ribbon tab, select the Grids > Solid Grid tool in the Structure group.

   The corresponding Snap Node/Beam, Snap Node/Plate, or Snap Node/Solid dialog opens. If an existing grid is selected in this dialog, this grid is displayed in the view window.

2. Click Import.
   - The Import Options dialog opens.

3. Select STAAD.Pro Grid (.grd) and then click OK.
   - An Windows Open dialog opens.

4. Select the STAAD.Pro grid file and then click Open.
   - The grids in the file are added to the Snap Node/Beam dialog.

M. To import a DXF file as a grid

To import an AutoCAD® DXF file to use as a drawing grid, use the following procedure.

The straight lines within a DXF file can be imported as a grid. Complex shapes and curved lines will not be imported.

1. Either:
   - On the Geometry ribbon tab, select the Grids > Beam Grid tool in the Structure group.
or
On the **Geometry** ribbon tab, select the **Grids > Triangular Plate Grid** tool in the **Structure** group.

or
On the **Geometry** ribbon tab, select the **Grids > Quad Plate Grid** tool in the **Structure** group.

or
On the **Geometry** ribbon tab, select the **Grids > Solid Grid** tool in the **Structure** group.

The corresponding **Snap Node/Beam, Snap Node/Plate, or Snap Node/Solid** dialog opens. If an existing grid is selected in this dialog, this grid is displayed in the view window.

2. Click **Import**.
   The **Import Options** dialog opens.

3. Select the **DXF** option and click **OK**.
   The **DXF Import** dialog opens.

4. In the File Name field, either:
   - type the file path and file name of the DXF file
   - or
   - click [...] locate the file using the **Open DXF File** dialog
   The layers and line data is read and displayed graphically.

5. Select the layer number containing the line data you want to use from the **Layers** drop-down list.
   The lines are displayed in the list view and graphically.

6. (Optional) Uncheck any lines you do not want to include in the imported grid.
   The lines are hidden in the view window.

7. (Optional) Type a **Name** for the imported grid.
   By default, the name of the DXF file is used.

8. Select the **Unit** to use for the grid.

   **Tip:** The drop-down list contains most common length units. Use the up or down arrows to navigate through the options.

9. Click **OK**.
   The Imported Grid pop-up dialog opens.

10. (Optional) Specify the **Angle of Plane** and **Grid Origin** to orient the grid.

11. Click **OK**.
When you save your model, the imported DXF grid will be saved to the corresponding STAAD.Pro grid file (file extension .grd).

**Related Links**
- *Imported Grid dialog* (on page 2900)
- *DXF Import dialog* (on page 2899)

## M. Beams

### M. To set attributes for new beams

To specify a set of attributes to automatically assign to new beams, use the following procedure.

For member profiles and member end releases, you will need to add the corresponding specification to your model before you can select it for most attribute sets.

**Tip:** The predefined attribute sets have some pre-populated values that will be added to your model automatically, though.

In STAAD.Pro, you will often assign attributes (beta angle, end offsets, material, profile, etc.) after you have placed a member. However, you can also assign attribute sets when creating new members by defining these ahead of time. STAAD.Pro allows you to create and edit named attribute sets so you can easily change between attribute sets.

**Note:** This feature is not active by default so new members will not have any attributes assigned initially.

1. On the **Geometry** ribbon tab, select **Add Beam > Set New Beam Attributes** in the **Beam** group. The **Define Member Attributes** dialog opens.
2. Select an attribute set in the list. **Tip:** There are several named sets which can be used or you can simply use the default “(None)” set.
3. Check the box for each attribute you want assigned. **Note:** This option can only be checked for one attribute set.

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member Property</td>
<td>Check to <strong>Assign Profile with Material</strong> and then select the profile and associated material from the drop-down list.</td>
</tr>
<tr>
<td>Beta Angle</td>
<td>Check to <strong>Assign Member Rotation</strong> and then select the option for rotation from the list. Type in the <strong>Angle in Degrees</strong> if you select that option.</td>
</tr>
<tr>
<td>Member Release</td>
<td>Check to <strong>Assign to Member Start</strong> and <strong>Assign to Member End</strong> as needed. Select a end release specification from the drop-down list for each end you have added a member release.</td>
</tr>
</tbody>
</table>

4. Click **Save**. The attribute set is updated with the new specification selections.
5. Check the option to **Assign these attributes when creating members**.
   **Note:** This option can only be checked for one attribute set.
6. Click **OK**.
M. To add beams by drawing on a grid

1. On the Geometry ribbon tab, select the Grids > Snap Grid Beam tool in the Structure group.

   The Snap Node-Beam dialog (on page 2895) opens and the current grid is displayed in the active View window.

   **Tip:** The dialog and grid are automatically opened if you select Start adding beams in the New model wizard.

2. (Optional) Either:
   - click Edit to edit the currently selected grid
   - or
   - click Create to create a new custom grid (radial and irregular grids must be custom)

3. (Optional) Set the Snap to existing nodes too

   **Note:** Existing nodes have precedence over grid intersections, in order to prevent creating duplicate nodes (on page 889).

4. If it is not already active, click Snap Node/Beam in the Snap Node/Beam dialog.

5. In the Active view window, click any grid intersection or existing node (if the option was set) to select the start point of a new beam member.

   A cross icon "snaps" to the nearest grid line intersection or existing node to the mouse pointer. This indicates the tentative point for placing a beam end. Once the start end is selected, a red circle indicates the start point and a line is rubber-banded to the snap point nearest the mouse pointer.

6. Click the end point of the beam.

7. Repeat steps 5 and 6 until you are finished drawing beams.

8. To stop drawing beams, either:
   - press <Esc>
   - or
   - click Snap Node/Beam in the Snap Node/Beam dialog

M. To add beams with new nodes

To add beams between existing nodes or along beams by generating new nodes, use the following procedure.

1. On the Geometry ribbon tab, Add Beam tool in the Beam group.

   The mouse pointer changes to the Add Beams cursor (➕).

2. Click on any point along the first beam where the starting node of the new beam will lie.

   If the program detects you are clicking on an existing node, skip to Step 6.

   If an existing node is not present at that point, a dialog box will prompt for a new node to be created.
3. Click **Yes**. The **Insert Nodes into Beam #** dialog opens. The value in the **Distance** edit box is the exact location clicked on from the start of the beam.

4. (Optional) Edit the **Distance** value as necessary.

   **Tip:** Other nodes can also be generated along the beam as well. For more information on how to use the **Insert Node** dialog box, please refer to [M. To insert a node in a single member](on page 697).

5. Click **OK**

   A new node is created on the selected beam and a the new beam member is “rubber banded” to this point.

   **Tip:** If the new node input is not within a close proximity of the point clicked on the screen (that which resulted in the dialog opening), no “rubber band” line will be shown. In this case, simply click on the new node to start the creation of the beam.

6. Click a point along another beam or an existing node to specify the end point of the new beam.

   If this is not at an existing node, refer to steps 2 through 5 for inserting a new node into a beam.

7. To stop adding beams:

   Press `<Esc>`

   or

   select any tool

**Related Links**

- [TR.12 Member Incidences Specification](on page 2428)

**M. To add beams from mid-points**

1. On the **Geometry** ribbon tab, select the **Add Beam > Add Beam Between Mid-Points** tool in the Beams group.

   ![Add Beams cursor](image)

   The mouse pointer changes to the Add Beams cursor (🔗).

2. Click on any point along the first beam where the starting node of the new beam will lie.

   A new beam member is "rubber banded" to this point.

3. Click on any point along the second beam where the end node of the new beam will lie.

   New nodes are created on the selected beams at their mid-points and a the new beam member connects these two new points.

4. Repeat steps 1 through 3 to continue adding beams.

5. To stop adding beams:

   Press `<Esc>`

   or

   select any tool
M. To add beams perpendicular to existing beams

1. On the **Geometry** ribbon tab, select the **Add Beam by Perpendicular Intersection** tool in the **Beam** group.

   The mouse pointer changes to the Add Beams cursor (_attach_point).

2. Click on any point along the first beam where the starting node of the new beam will lie. A new beam member is "rubber banded" to this point.

3. Click on any point along the second beam where the end node of the new beam will lie. A new node is created on the selected beam at the at the calculated perpendicular point and a the new beam member is added.

4. Repeat steps 2 and 3 to continue adding beams.

5. To stop adding beams:
   - Press `<Esc>`
   - or
   - select any tool

M. To add a curved beam

To add a curved beam between two existing nodes, use the following procedure.

Curved members may only be created between two existing nodes. Any non-tapered cross-section is permitted for curved members. The internal angle subtended by the arc must be less than 180 degrees.

**Note:** The design of curved members is *not* supported.

1. On the **Geometry** ribbon tab, select the **Add Beam > Add Curved Beam** tool in the **Beam** group.

   **Tip:** This tool is contained on a drop-down list below the **Add Beam** tool.

   The pointer changes to the Add Curved Beam cursor (_attach_point).  
2. Click on the start and end nodes, respectively.
   - The **Curve Beam Properties** dialog opens.
3. Type the **Radius of Curvature** (in current units of length) and the **Gamma Angle** (in degrees) values.

   The gamma angle, \( \gamma \), is the angle between the member local Y axis and the plane of the circular arc of the member.

4. Click **OK**.
   The curved beam will be added to the model.

5. Repeat steps 2 through 4 to add more curved beams.

6. To stop adding beams:
   - Press `<Esc>`
   - or
   - select any tool

**Related Links**
- *G.6.8 Curved Members* (on page 2329)
- *TR.20.8 Curved Member Specification* (on page 2475)

**M. To split a beam at selected node(s)**

1. Select one or more nodes located along a beam.
2. On the **Geometry** ribbon tab, select the **Break Beams at Selected Nodes** tool in the **Beam** group.
A message box appears which displays the number of new beams created.

3. **Click OK**
   The dialog closes.

### M. To stretch a member

1. Select one or more members which are to be extended.
2. On the **Geometry** ribbon tab, select the **Geometry > Stretch Selected Member(s)** tool in the **Beam** group.

   The **Stretch Member(s) dialog** (on page 2908) opens.

3. **(Optional)** Click the drop-down list in the **Select member(s)** and then uncheck any members you do not wish included in the stretch operation.

4. Specify the method of stretch and required parameters:
   - Select **To a point** and then type the coordinates of an arbitrary point
     
   - Select **Through a distance** and then select the member end and distance to extend to
     
   - Select **To an existing node** and then select the node number
     
     **Tip:** Click the tool to graphically select a node.

   or

   - Select **To an existing member** and then select the member number
     
     **Tip:** Click the tool to graphically select a member.

5. **Click OK**

### Related Links

- **Stretch Member(s) dialog** (on page 2908)

### M. To merge two or more members

To merge two or more members into a single member, use the following procedure.

1. Select the beams to be merged.
2. On the **Geometry** ribbon tab, select the **Merge Selected Beams** tool in the **Beam** group.
The Merge Selected Beams dialog opens.

3. Specify the member properties the merged member will have.
4. Click Merge.

Related Links
- Merge Selected Beams dialog (on page 2909)

M. To renumber selected beams

To renumber the currently selected members, use the following procedure.

1. Select the beams to be renumbered.
2. On the Geometry ribbon tab, select the Renumber Beams tool in the Beam group.

A message dialog opens to warn you that renumbering may not be undone.

3. Click Yes to confirm you wish to proceed.
   The Renumber dialog opens.

4. (Optional) Specify a Start number from
5. (Optional) Select the New Numbering Order
6. (Optional) Select the sort criteria to be used and add them to the Selected Sort Criteria
7. (Optional) Reorder the items in the Selected Sort Criteria
   For example, select the Z Coordinate and click Move Up twice so the order from top to bottom is: Z Coordinate, X Coordinate, Y Coordinate. This instructs the program to renumber beams parallel to the Y axis until no more beams are in that line. It then skips to the next selected entity along the X axis and renumbers those, again along the Y axis. Once it has reached the end of selected beams parallel to the X axis, it then skips to the next selected entity parallel to the Z axis and repeats the previous procedures.
8. Click Accept.
   A message dialog opens with the status of the renumbering operation.
9. Click OK to dismiss the message dialog.
   The selected beams are renumbered following the indicated pattern.

Related Links
- Renumber dialog (on page 2904)

M. Physical Members

Physical members are composites of one or more analytical members which represent the true physical geometry of a member in a structure.

Note: Physical members use the nomenclature PMEMBER in the STAAD input file.

About Physical Members

Beam objects connecting two nodes in a structural model are referred to as analytical beams as they are the fundamental, one-dimensional object processed by the STAAD engine during analysis. While necessary for this
purpose, they are often difficult elements to use for design as they do not possess the same geometry of a 
physical structural element. For example, a girder is not physically broken where a beam frames, but a node 
must be placed to represent the connection in the model. The design of this girder is then complicated as the 
bracing conditions and other parameters of design can not be directly interpreted by the program.

In this case, STAAD.Pro has the facility to use group one or more analytical beam objects into physical members 
for the purpose of design. This concept is used in both the Steel Design and Concrete Design workflows. 
Additionally, physical members can be created in the modeling mode for the purpose of integrated steel design.

Related Links
- TR.16.2 Physical Members (on page 2443)
- M. To assign catalog section to physical members (on page 742)

M. To manually form physical members

To form a physical member for a selected set of analytical members, use the following procedure.

The following are required to form a physical member:
- analytical members must be interconnected
- analytical members are collinear
- the local axes of the component members should be identical (i.e., the x, y, and z are respectively parallel and 
in the same sense)
- a single analytical member may not be in more than one physical member

1. Select one or more beams which represent a single physical member.

   **Note:** Component analytical members must meet the criteria for physical members.

2. Either
   - on the **Beam** ribbon tab, select the **Form Member** tool in the **Model** group
   - or
   - on the **Geometry** ribbon tab, select the **Form Member** tool in the **Physical Member** group (Analytical 
   Modeling workflow)
   - or
   - on the **Member Design** ribbon tab, select the **Form Member** tool in the **Physical Member** group (Steel 
   Design workflow)
   - or
   - right-click and select **Form Member** from the pop-up menu.

The physical member (i.e., M#) number is displayed on each component analytical member and the physical 
member is added to the **Physical Member** table.
M. To automatically form physical members

To have the program automatically detect and form physical members within a selected set of members, use the following procedure.

1. Select a set of beams that you want formed into physical members.
2. Either:
   - on the Geometry ribbon tab, select the AutoForm Member tool in the Physical Members group (Analytical Modeling workflow)
   
   ![AutoForm Member Tool](image)

   or
   
   - on the Member Design ribbon tab, select the AutoForm Member tool in the Physical Members group (Steel Design workflow)

The program applies the rules of forming physical members to detect which members should be grouped to a single physical member.

M. To automatically generate physical member restraints

Use the following procedure to automatically generate top and bottom flange restraint conditions for the selected physical members.

**Note:** The PBRACE command generated by this menu item is valid only for [AS 4100 (Australian) steel design](on page 1649).

1. Create a STAAD.Pro model with steel members.
2. Either:
   - on the Analysis and Design ribbon tab, select the Design > Steel Design tool in the Design group
   or
   - select the Design page

   The Steel Design - Whole Structure dialog opens.

3. Form one or more physical members in the model.
4. Select AS4100 as the Current Code.
5. Select one or more of the physical members.
6. On the Utilities ribbon tab, select the Physical Member Restraints tool in the Geometry Tools group.

   The PBRACE commands are added for the current design parameter set with the tag (Physical).

   **Note:** Only top or bottom flange restraints can be described using a single PBRACE command. Therefore, this command generates to lines of PBRACE commands for each physical member.
M. To manually add physical member restraints

To manually add physical member restraints for assignment to physical members, use the following procedure. Manual restraints apply to steel physical members which will be designed per AS 4100.

**Tip:** The restraint details can be automatically generated using a new **Member Restraints** item found in the **Utilities** ribbon tab in the **Geometry Tools** group.

1. In the Analytical Modeling workflow page controls, select the Design page.
2. Select **AS4100** as the **Current Code** in the **Steel Design - Whole Structure** dialog.
3. Click **Define Parameters**.
   - The **Design Parameters** dialog opens.
4. Select the **PBRACE** parameter.
5. For each brace point, specify a fraction of the total physical member length in the **Brace Location** cell.

![Edit Design Parameter](image)

6. Select the type of restraint present for the top or bottom flange.
Only top or bottom flange restraints can be described using a single \texttt{PBRACE} command. If both top and bottom bracing is added in the Design Parameters dialog, this will generate two new command lines in the input file.

7. Click \texttt{Add}.
   The new command appears in the tree as a child element of the current design parameter set.

8. Click \texttt{Close}.
   The design parameters are marked with an \texttt{?} icon. This indicates that the need to be assigned to members.

9. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

The \texttt{PBRACE} specification is tagged as (Physical), and therefore can only be assigned to physical members (\texttt{PMEMBER} groups).

**Related Links**

- \textit{D2.B.12.6 Physical Member Restraints Specification} (on page 1653)

**M. To delete a physical member**

1. Select the \textbf{Member Number ID} or any component Beam Number ID in the \textbf{Physical Member Table}.
2. Right click to display the pop-up menu.
3. Select \texttt{Delete}.
   A confirmation dialog opens.
4. Select \texttt{Yes}.

   \textbf{Note}: Only the physical member group assignment is removed. The component analytical beams are not deleted.

   \textbf{Caution}: This cannot be undone.

**M. Plates**

**M. To set new plate attributes**

To specify a set of attributes to automatically assign to new beams, use the following procedure.

1. On the \textbf{Geometry} ribbon tab, select the \texttt{Add Plate > Set New Plate Attributes} tool in the \textbf{Plate} group. The \textbf{Define Plate Object Property} dialog opens.
2. Select the check box for each of the plate object properties you want to use and then select the property value you want to use from the drop-down list:

   \begin{tabular}{|l|l|}
   \hline
   \textbf{Option} & \textbf{Description} \\
   \hline
   \textbf{Property} & Check the \textbf{Use Property} option and then select a defined plate or surface thickness property from the drop-down list. \\
   \textbf{Material} & Check the \textbf{Use Material} option and then select a material definition name from the drop-down list. \\
   \textbf{Release} & Check the \textbf{Use Release} option and then select a plate release specification from the drop-down list. \\
   \hline
   \end{tabular}
3. (Optional) To create a new specification to use, click the associated Create button with the use option checked.
   The corresponding dialog opens to add a property, material, or release. Once you add this specification, the new specification is available from that drop-down list for use.
4. Check the option to Assign these attribute when creating new plates.
5. Click OK.

Related Links
• Define Plate Object Property dialog (on page 2916)

M. To draw plates connecting existing nodes

To draw triangular or quadrilateral plate elements connecting existing nodes, use the following procedure.
This procedure is used to manually add plate elements to your model. STAAD.Pro also has tools to automatically generate a finite element mesh with a surface area.

1. On the Geometry ribbon tab, select one of the following tools in the Plates group:
   Use this tool... To...
   Add Quad Plate create four-node plate elements
   Add Triangle Plate create three-node plate elements

The mouse pointer changes to either the Add Quad Plate cursor ( ) or Add Triangle Plate cursor ( ), respectively.
2. Click on any node that will form the first corner of the plate element.
   A line is “rubber banded” to the cursor from this node. This represents the first edge of the surface.
3. Click on the subsequent nodes to form the plate vertices in either a clockwise or counterclockwise order.
   The plate shape is “rubber banded” to the cursor as you move the mouse pointer to give you a preview of the plate shape.

   Note: Plate vertices should lie within a flat plane. Refer to “ M. To check for warped plates (on page 887)” to verify that plates are planar.

   The plate is added to the model.
4. Repeat steps 2 and 3 to add more plates.
5. To stop adding plates:
   press <Esc>
   or
select any tool

**Related Links**
- *TR.13.1 Plate and Shell Element Incidence Specification* (on page 2431)

**M. To add a plate bounded by beams**

To create an “infill” plate by selecting exiting members as plate edges, use the following procedure.

**Note:** The beams to be used as bounding edges of an “infill” plate must meet the following requirements:
- three or four beams that form a closed polygon
- the beams cannot extend past the vertex nodes of the closed polygon
- the beams must lie within a plane

<table>
<thead>
<tr>
<th>1. Select the beams you want to use as bounding edges of the plate. You must select three or more beams.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Note:</strong> Use the <strong>Beam Cursor</strong> tool in the <strong>Selection</strong> group on the <strong>Geometry</strong> ribbon tab.</td>
</tr>
<tr>
<td>2. On the Geometry ribbon tab, select the <strong>Add Plate &gt; Create Infill Plates</strong> tool in the <strong>Plates</strong> group.</td>
</tr>
</tbody>
</table>

A message dialog indicates that the plate was successfully generated.

**M. To generate plate mesh from corner nodes**

To generate a finite element mesh by selecting corner nodes, use the following procedure.

You must have three or more existing nodes that lie in a plane in your analytical model.

This procedure is used to immediately generate a mesh within a selection of vertices. If you would like to investigate changing parameters prior to generating a mesh or to add openings with the mesh, then you may want to use the procedure a *parametric mesh model* (on page 656) instead.

<table>
<thead>
<tr>
<th>1. On the <strong>Geometry</strong> ribbon tab, select the <strong>Generate Mesh &gt; Create Mesh</strong> tool in the <strong>Plate</strong> group.</th>
</tr>
</thead>
<tbody>
<tr>
<td>The mouse pointer changes to a mesh cursor.</td>
</tr>
<tr>
<td>2. Click the nodes that will form the vertices of the mesh area, in either a clockwise or counter clockwise order.</td>
</tr>
</tbody>
</table>

**Notes:**
- The order of the nodes should be in either a clockwise or counterclockwise direction.
- The nodes must for a polygonal area (i.e., they edges cannot cross).
- The nodes must lie in the same plane.

| 3. When you have select the corner nodes, either: |
select the first node again
or
press <Esc>
The Choose Meshing Type dialog opens.

4. Select the type of meshing to use and then click OK

   **Polygonal Meshing**    Only triangular elements will be created
   **Quadrilateral Meshing** This will create principally four sided elements, but where the geometry dictates, some places may require triangular elements.

   The Define Mesh Region dialog opens.

5. Review the mesh vertices:
   a. Click the Add New Row tool to add a new vertex.
   b. Click the Delete Row tool to remove the currently selected row.
   c. Select the Holes tree entry to add openings to the mesh.

6. Click OK.
The specified area within the vertices is meshed.

M. Parametric Models

STAAD.Pro allows you to model surfaces by automatically generating a mesh from a set of parameters. You can preview the final meshed state of the wall, slab, or panel before deploying the meshed entity into the remainder of the model. This way you can visually examine several trials or prototypes to find the most suitable alternative.

**Tip:** In past versions of STAAD.Pro, facilities for mesh generation have been available through the Geometry menu of the main program window and through the Structure Wizard utility which too is accessible through the Geometry menu. While these facilities continue to be available, an enhanced mesh generation tool has been added to the Geometry page.

This method offers some advantages over other methods of mesh generation in STAAD.Pro:

a. It enables you to preview the final meshed state of the wall, slab or panel before deploying the meshed entity into the remainder of the model. Several trials or prototypes can visually examined before committing to the most suitable one. This greatly minimizes the inconvenience of ending up with an undesirable meshed panel.

b. During the process of meshing, the program will automatically take into consideration existing nodes that lie within the boundary of the panel, but are not chosen as corner nodes of the panel. You simply have to define the panel boundary. The remainder of the nodes will be automatically considered by the program as “control
For generating a finite element mesh, use the following procedure.

1. Select the **Parametric Models** tool in the Plate group on the **Geometry** ribbon tab.

The **Parametric Models** dialog opens. The structure is displayed as a series of dashed lines to signify this mode can be used to experiment with various settings.

2. Click **Add** in the **Parametric Models** dialog.

The **New Mesh Model** dialog opens.

3. Enter a name for the new model (or accept the default).

4. Click **OK**.

The mouse pointer changes to the add surface cursor.

5. Click on the points that form the corners of the panel boundary.

**Note:** Click the points in either a clockwise or counter-clockwise order.

**Tip:** If left the option selected for including nodes and beams in the boundary area, then you can skip over intermediate points lying and a straight line segment, as the program will detect these and add them as mesh nodes automatically.

When the first point is clicked a second time to close the loop, the **Mesh Parameters** dialog opens.

6. Specify the desired meshing parameters and click **OK**.

A dialog opens asking if you wish to add openings to the mesh.

7. (Optional) Click **Yes**

You are asked if the opening is rectangular or circular. Select an option and click the **OK** button to draw the opening on the meshed surface. Refer to the following tasks for additional information on adding openings.

**Tip:** Openings can be added later if needed by clicking the **No** button when asked to add these.

The new mesh model is displayed as an overlay on the structure with control points highlighted. The details of the mesh parameters are available in the Parametric Models dialog.

**Note:** You may edit and refine your mesh by varying the parameters as necessary until the resulting finite element is satisfactory for your model requirements.
8. Once you are satisfied with the surface mesh, click **Merge Mesh** in the Parametric Models dialog to commit it to the STAAD.Pro input file.

**Tip:** To delete a parametric mesh, opening, density line, or density point, select that entry within the **Parametric Models** dialog and click **Delete**.

**Related Links**
- Parametric Models dialog (on page 2910)
- Mesh Parameters dialog (on page 2912)
- New Mesh Model dialog (on page 2912)

**M. To create a polygonal opening in a mesh model**

1. In the **Parametric Models** dialog Mesh Models list, select the leaf called **Openings** under the mesh model to which you wish to add an opening.
2. Click **Add**.
   - The Select Opening Type dialog opens.

3. Select the **Polygonal Opening** option and then click **OK**
   - The drawing will change to display just the mesh region. The right hand side of the screen will display a grid with the grid line settings and spacing.
   **Tip:** The various viewing options and zoom options on top of the screen can be used to obtain a convenient view of the panel.
4. (Optional) Edit the grid positions using the **Snap Node/Panel**
5. Click the points inside the panel to define the corners of the polygonal hole.
   **Note:** The cursor displays a blinking cross (+) at grid intersections. This blinking cross marks the point on the grid at which nodes of the hole boundary may be set. Click points around the mesh boundary in either a clockwise or counter-clockwise order, clicking on the first point a second time to close the loop.
   - The mesh is regenerated taking into consideration the hole.
   **Tip:** To delete a parametric mesh, opening, density line, or density point, select that entry within the **Parametric Models** dialog and click **Delete**.
M. To create a circular opening in a mesh model

**Note:** In case an opening is required on a part of the structure that is already meshed with plate elements, the existing plates have to be deleted first, and then, the plates with the opening can be modeled as a whole using this feature.

1. In the **Parametric Models** dialog Mesh Models list, select the leaf called **Openings** under the mesh model to which you wish to add an opening.
2. Click **Add**.

   The **Select Opening Type** dialog opens.

3. Select the **Circular Opening** option and then click **OK**.

   The drawing will change to display just the mesh region. The right hand side of the screen will display a grid with the grid line settings and spacing.

   **Tip:** The various viewing options and zoom options on top of the screen can be used to obtain a convenient view of the panel.

4. (Optional) Edit the grid positions using the **Snap Node/Panel**.

5. Click the point which will be the center of the circular opening.

   The cursor displays a blinking cross (+) at grid intersections. This blinking cross marks the point on the grid at which nodes of the hole boundary may be set.

6. Click a second point to define the radius of the circle.

   A dialog opens for defining the number of divisions around the circular opening.

7. (Optional) Change the number of divisions used around the circumference of the circular opening.

8. Click the **OK** button.

   The mesh is regenerated taking into consideration the hole.

   **Tip:** To delete a parametric mesh, opening, density line, or density point, select that entry within the **Parametric Models** dialog and click **Delete**.

M. To add a density line or point

1. In the Parametric Models dialog Mesh Models list, click on the leaf called **Density Lines** or **Density Points** under the mesh model to which you wish to add an opening.

2. Click **Add**.

   The drawing will change to display just the mesh region. The right hand side of the screen will display a grid with the grid line settings and spacing.
Tip: The various viewing options and zoom options on top of the screen can be used to obtain a convenient view of the panel.

3. (Optional) Edit the grid positions using the Snap Node/Panel
4. Either:
   Click the start and points of a density line.
   or
   Click a density point.
   The mesh is regenerated to account for the new density line or point
5. Repeat step 4 as many times as needed.
6. To stop adding density points or density lines, press the <Esc> key.

Tip: To delete a parametric mesh, opening, density line, or density point, select that entry within the Parametric Models dialog and click Delete.

**M. To define a slab/wall connection**

Use the following procedure to generate a mesh for a wall super-element when compatibility with a connected slab super-element is required.

A slab mesh must first be created. The slab mesh should include a density line representing the connecting edge of the wall.

Tip: Using beam elements is fast way to help generate meshed surfaces and ensure that the density line edge for the slab/wall connection is included when the slab is generated. These "construction" beams can be deleted once the surface elements have been successfully generated.

1. On the Geometry ribbon tab, select the Generate Slab/Wall Connection tool in the Structure group.

2. In the Active view window, click the corner nodes of the wall super-element in either a clockwise or counterclockwise order.

   **Note:** The first and second nodes must be the corner nodes which form the slab/wall connection. Failure to do so will open a warning dialog.

3. Continue clicking the remaining corner nodes to for the wall super-element. Close the polygon by clicking on the first node again.

   The Division Along Wall dialog opens.

   **Note:** The divisions along the horizontal side of the wall have already been decided as the number of points created along the wall base in the previously mesh slab.

4. (Optional) Specify a No. of divisions in extrusion direction
**Note:** The divisions along the horizontal side of the wall have already been decided as the number of points created along the wall base in the previously mesh slab.

5. Click **OK**  
The wall is meshed.

**Tip:** To verify the compatibility between plates, select the **Plate Tools > Check Improperly Connected Plates** tool in the **Geometry Tools** on the **Utilities** ribbon tab.

**Related Links**
- *EX. Meshed Wall-Slab Connection* (on page 4909)

**M. Advanced Surface Mesher**

A preprocessing module for creating finite element meshed surfaces for use with the analysis and design program STAAD.Pro.

**M. To launch the Advanced Mesher program**

1. Either:
   - In the **Bentley Engineering** program group on the Windows **Start** menu, click the **Mesher** icon.
   - or
   - In Windows Explorer, double-click a **SurMesh** file icon (file extension .smh).
   - or
   - Drag an .smh file icon from Windows Explorer and drop it on the **Mesher** icon.
   - or
   - In Windows Explorer, double-click the icon for the program file, **SurMesh.exe**. This file can be located in C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\Mesher in a typical STAAD.Pro installation.

**M. Basics**

Surfaces are created by specifying an external contour, which can be perforated by one or more openings. You can define locations that require specific node positions, such as column positions and areas of higher mesh density. You then select the meshing routine to use and the Advanced Surface Mesher will create a meshed surface that can be refined further and transferred to STAAD.Pro for inclusion in a STAAD.Pro structure.

**M. Example**

Use this procedure to create an example meshed solid which can then be analyzed in STAAD.Pro.
This tutorial utilizes the basic features of the program in order to familiarize you with the process of creating a simple model. Not all tools or options are presented here.

1. Select the **New** tool on the Model toolbar
   An empty file is opened.
2. Select the **Overall Dimensions** tool.
   The Overall dimension dialog opens.
3. Specify a **Width** of 40 m and **Height** of 40 m and then click **OK**.
   Leave the plane as XZ.
   The dimension limits are drawn in the View window.
4. Select the **Grid Options** tool.
   The Grid settings dialog opens.
5. Specify an X spacing of 5 m and a Z spacing of 5 m and then click **OK**.
   The drawing grid is displayed inside the overall dimensions within the View window.
6. If the **Snap to Grid** tool is not set, do so.

   **Tip:** When a tool is set, its icon is shown as depressed in the toolbar and in the menu.

7. Select the **External Contour** tool and click any vertex point as shown in the following figure and table.

<table>
<thead>
<tr>
<th>X (m)</th>
<th>Z (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>35</td>
<td>5</td>
</tr>
<tr>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>15</td>
<td>35</td>
</tr>
<tr>
<td>15</td>
<td>20</td>
</tr>
<tr>
<td>5</td>
<td>20</td>
</tr>
</tbody>
</table>

*Figure 98: External contour*
8. Continue clicking adjacent vertex points in either clockwise or counterclockwise order until the rubber-banded shape looks like the final shape desired; then double-click the final vertex point.

**Tip:** You can right-click to undo the previous data point if necessary.

The external contour shape is shaded gray.

9. Select the Internal Contour tool and click any vertex point of the opening shown in the following figure and table.

![Figure 99: Opening in the surface](image)

<table>
<thead>
<tr>
<th>X (m)</th>
<th>Z (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>10</td>
</tr>
<tr>
<td>30</td>
<td>10</td>
</tr>
<tr>
<td>30</td>
<td>30</td>
</tr>
<tr>
<td>25</td>
<td>30</td>
</tr>
<tr>
<td>15</td>
<td>20</td>
</tr>
<tr>
<td>5</td>
<td>20</td>
</tr>
</tbody>
</table>

10. Continue clicking adjacent vertex points in either a clockwise or counterclockwise order until the rubber-banded shape looks like the final shape desired; then double-click the final vertex point. The shading is removed inside the inner contour shape to represent the opening.

**M. Technical Reference**

This section details the engineering theory and methods behind the program.

The Advanced Mesher uses the Delanay meshing principles to create a triangular mesh using vertices and density settings specified by the user. The basic Delanay routine can be modified by three options known as the Triangulation Methods. On top of this there is an additional Parallel Grid quad option that can be used on surfaces that are defined with edges between vertices that lie purely in the X and Z axes.
**Triangulation Methods**

The Advanced Mesher uses three types of meshing routines:

1. Triangulation using the defined vertices only
2. Triangulation without sub-dividing the contours, but creating additional internal elements.
3. Triangulation sub-dividing the contours as well as creating internal elements.

Method A) uses the Delaunay triangulation – DT.

Methods B) and C) use the constrained Delaunay triangulation – CDT on PSLG (planar straight line graph).

The points created by the meshing routine that were not defined by the user in methods B) and C) are referred to as Steiner points or Free Nodes. When creating the triangles in method B) and C) the program ensures that only acute angles or obtuse angles are created in the mesh. Additionally, with method C) the user specified triangulation step is used to limit the maximum length of the side of a triangle or the area of the triangle. Note that there is a direct relationship between the length of the longest side of a triangle and its area.

**Definitions**

- **height**: the maximum distance to a vertex from a longest edge.
- **aspect ratio**: the ratio of the longest length to its height.
- **u**: the minimum internal angle.

Therefore, the aspect ratio \( > \frac{1}{\sin(u)} \) and \( \leq \frac{2}{\sin(u)} \)

Also if the step \( \leq L \), Area of triangle \( \leq \frac{L^2}{4 \sin(u)} \)

![Figure 100: Triangle element nomenclature](image)

**triangulation density**

Defined for each triangulation vertex. The triangulation density is the ratio between the maximum length of the lines emanating from the vertex, to the triangulation step. Where the length has been limited by the triangle area (or length), then the triangulation density of a given vertex is the ratio of the area of the largest triangle connected to the vertex to the triangulation parameter. Therefore, the triangulation density never exceeds one (unity).

Note that the program allows the user to specify triangulation density for:

- **a.** Nodes (thickening nodes)
- **b.** Lines (thickening lines), specified density applies to all vertices along the line.
- **c.** Regions (thickening region), specified density applies to all vertices that occur within the region including the edge.

**Note:** Also, if a vertex is covered by more than one density setting — such as a density line and a density region — then the maximum density setting will be used for either situation.
The formation of triangles

Consider the following plane surface with an internal opening shown in the following figure. It is defined by two contours, an external and a single internal contour. Straight lines between vertices define the contours. Vertices of contours are indicated with a red circle.

![Figure 101: Plan surface with opening](image1)

Triangulation is defined as the representation of the plane surface with a set of united, inter-connected (i.e., sharing edges), but not overlapping triangles.

![Figure 102: Simple triangulation](image2)

The vertices of the triangles are referred to as triangulation nodes. In the above simple triangulation, only the vertices defined by the contours are used. However, you can specify additional nodes inside the surface as shown in the following figure. The supplementary triangulations nodes assigned are indicated with a cyan circle.
If you choose Triangulation Method B) or C), then additional nodes may be generated by the program. These nodes are known as Steiner’s points, or free nodes, and are indicated in the following figure with a magenta circle.

![Triangulation with intermediate nodes](image1)

*Figure 103: Triangulation with intermediate nodes*

After the initial meshing process, these free nodes can be modified to suite a specific requirement such as maximising the minimum angle on the triangles, using the Smoothing tool.

**M. Technical Notes**

Aspect Ratio, Triangulation Step, and Area Limitation

![Aspect Ratio Diagram](image2)

The aspect ratio (AR) is defined as:
\[ AR = \frac{L_1 + L_2}{h} = \frac{1}{\tan(\theta)} + \frac{1}{\tan(\alpha)} \geq \frac{2}{\tan(\theta)} = \frac{2 \cos(\theta)}{\sin(\theta)} \geq \frac{1}{\sin(\theta)} \]

Of the two angles shown the diagram, \( a^2 q \) and monotonic decrease of function \( 1/\tan \) are used here. In addition, since angle \( q \) is the minimum, therefore \( 2\cos(q)^2 \)

By analogy:

\[ AR = \frac{L_1 + L_2}{h} = \frac{1}{\tan(\theta)} + \frac{1}{\tan(\alpha)} = \frac{\sin(\theta + \alpha)}{\sin(\theta)\sin(\alpha)} \leq \frac{\sin(\theta)\cos(\theta) + \cos(\theta)\sin(\alpha)}{\sin(\theta)\sin(\alpha)} \]

It means that to prove the above estimation it is necessary to demonstrate that

Since \( 90^o a^2 q \), then

It remains to note, that .

Since area of a triangle is equal to \((L_1+L_2)/2\) then it leads to, that is the above estimation for area, when angle \( q \) is constrained from below, results in to the above estimation of triangulation step.

Moreover, instead of estimation using the more accurate estimation we obtain

Jacobian

\[ \begin{align*}
\text{Coordinate element} & \quad W \\
\text{Ideal element} & \quad AW^{-1} \\
\text{Real element} & 
\end{align*} \]

Matrix \( A \) (referred to as the Jacobian matrix) transfers points of the coordinate triangle to the real triangle. Matrix \( W \) transforms coordinate triangle into equilateral (ideal) triangle. Therefore matrix \( S = AW^{-1} \), which transforms ideal element into real one, defines deviation of real element from ideal one.

For any triangle, the free vertex which is located at the point \( x \), and two other vertices \(- xi, xj \), the Jacobian is equal to

\[ \det (xi - x, xj - x) \]

Algorithms Used

A - triangulation - is a classical Delaunay triangulation (see, for example, [9]). For other cases a version of J.Rupert [2] algorithm is used.

Smoothening

Basic smoothening algorithm is described in reference [10].

Laplace

Node is moved to the point, which is averaging of locations \( M \) of the nearest nodes

\[ x_n = \frac{1}{M} \sum_{m=0}^{M-1} x_m \]

References


M. Application Window Layout

![Figure 105:](image-url)
**M. Menus**

Menus in the Advanced Surface Mesher module window.

**Note:** Some of the menu items are contextually sensitive to the currently selected mode (tab).

**M. File menu**

Contains items for creating, opening, and closing section data files, printing, and exiting the program.

**Table 23: File menu items**

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Opens an empty surface design file. If the currently loaded section has been modified, then the option is given to save it prior to commencing with the new design.</td>
<td>CTRL+N</td>
</tr>
<tr>
<td>Open</td>
<td>Opens the Load Model dialog, which is used to select an existing Surface Mesh data file (file extension .SMH). Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the <strong>Preview</strong> option is checked, then a preview of the surface can be displayed prior to opening the file. <strong>Note:</strong> Only a single surface mesh data file may be open at a time, so you will be prompted to save any changes made to the current file, if one is open, and it will be closed prior to opening a different file.</td>
<td>CTRL+O</td>
</tr>
<tr>
<td>Save</td>
<td>Saves any changes made to the current data file. <strong>Note:</strong> If the current file has not been previously saved, this has the same effect as selecting <strong>Save As</strong>...</td>
<td>CTRL+S</td>
</tr>
<tr>
<td>Save As...</td>
<td>Opens the Save data file dialog which is used to specify a file name and location for the section data file. Files are saved in the Surface Mesh data file (.SMH) format.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
<td>Same effect as selecting…</td>
</tr>
<tr>
<td>-------------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Import DXF</td>
<td>Opens the Import DXF dialog, which is used to select CAD files to import. Files in the drawing exchange format (file extension .DXF) may be used for import.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Files to be imported must only contain the definitions of the contours, which must be defined by closed lines or polylines.</td>
<td></td>
</tr>
<tr>
<td>Parametric Sections...</td>
<td>Opens the Parametric Section dialog which is used to quickly create a surface based on a section type and a number of key dimensions and insert that section as an external contour.</td>
<td></td>
</tr>
<tr>
<td>Send...</td>
<td>Opens an e-mail (using your default e-mail client) and attaches the current data file.</td>
<td></td>
</tr>
<tr>
<td>Recent File list</td>
<td>A list of the most recent four section data files. Select one to open that data file.</td>
<td></td>
</tr>
<tr>
<td>Quit</td>
<td>Closes the program. If any changes were made since the last time the data file was saved, a message dialog opens confirming you wish to save those changes.</td>
<td>ALT+F4</td>
</tr>
</tbody>
</table>

M. Edit menu
Contains items for

Table 24: Edit menu items in the Model mode

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo</td>
<td>Negates the last drawing operation. You can undo a series of modifications by repeatedly selecting Undo.</td>
<td>CTRL+Z</td>
</tr>
<tr>
<td>Redo</td>
<td>Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting Redo.</td>
<td>CTRL+Y</td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
<td>Same effect as selecting…</td>
</tr>
<tr>
<td>---------------------------------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Cancel</td>
<td>Cancels the current drawing operation.</td>
<td>Esc</td>
</tr>
<tr>
<td>Overall Dimensions …</td>
<td>Opens the Overall Dimensions dialog, which is used to specify the size of the workspace that will be required to create the section. The minimum it should be set to is the overall dimensions of the section but can be set larger.</td>
<td></td>
</tr>
<tr>
<td>Draw external contour</td>
<td>Used to set or clear the external contour drawing mode, which defines the outside of the section.</td>
<td></td>
</tr>
<tr>
<td>Locate circular external contour</td>
<td>Opens the Radius of the external contour dialog, which is used to specify a radius value for a parametrically defined, circular external contour.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> This menu item is only active if no other external contour has been defined.</td>
<td></td>
</tr>
<tr>
<td>Draw internal contour</td>
<td>Used to set or clear the internal contour mode, which is used for creating internal openings within an external boundary.</td>
<td></td>
</tr>
<tr>
<td>Copy inner contour &gt; &lt;list&gt;</td>
<td>Used to window either a rectangle or polygonal area to select one or more internal contours for copying. Copies can be placed manually or by specifying offset distances.</td>
<td></td>
</tr>
<tr>
<td>Multiple copy of inner contour &gt; &lt;list&gt;</td>
<td>Used to window either a rectangle or polygon area to select one or more internal contours making multiple copying. Copies can be placed manually or by specifying offset distances and a number of copies.</td>
<td></td>
</tr>
<tr>
<td>Delete internal contour</td>
<td>Used to set or clear the delete internal contour drawing mode, which is used to remove internal contours by clicking anywhere in their boundary.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
<td>Same effect as selecting…</td>
</tr>
<tr>
<td>---------------------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Insert circular hole, center, and radius</td>
<td>Used to set or clear the circular hole drawing mode, which is used to define the center of a hole (e.g., a parametrically defined, circular inner contour) and radius graphically.</td>
<td></td>
</tr>
<tr>
<td>Insert circular hole with defined radius</td>
<td>Used to set or clear the circular hole with defined radius drawing mode, which is used to define the center of a hole graphically and then specify an exact radius.</td>
<td></td>
</tr>
<tr>
<td>Parametric hole …</td>
<td>Opens the Parametric holes dialog, which is used to define the location and dimensions of either a circular or rectangular inner contour.</td>
<td></td>
</tr>
<tr>
<td>Radius corner …</td>
<td>Used to set or clear the smooth corner drawing mode, which is used to select a corner to fillet and then opens the Smooth radius dialog to specify a radius value for the rounded corner.</td>
<td></td>
</tr>
<tr>
<td>Move vertices</td>
<td>Used to window either a rectangular or polygonal area to select one or more vertices to move. Vertices can be moved either manually or by specifying offset distances.</td>
<td></td>
</tr>
<tr>
<td>Edit vertices</td>
<td>Displays all vertex numbers in the View window and opens the Coordinates dialog which is used to specify precise coordinates of each vertex for a selected contour.</td>
<td></td>
</tr>
<tr>
<td>Delete vertices</td>
<td>Used to select a single vertex or window either a rectangular or polygonal area to select one or more vertices for deletion.</td>
<td></td>
</tr>
<tr>
<td>Coordinate center …</td>
<td>Opens the Coordinate Center dialog, which is used to specify shift the global coordinates to either an arbitrary location or to the center of gravity of the shape.</td>
<td></td>
</tr>
</tbody>
</table>
### Table 25: Additional edit menu items in the Triangulation mode

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create density points for contours</td>
<td>Used to add triangulation points on external or internal contour lines. As with the vertices for the contours, nodes for the triangulation will always be generated on these points. Density Points are displayed with a red dot.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If a vertex at either end of the line is moved, then the density point will be moved in such a way so that the point will remain in the same proportion along the line defined by the new vertices.</td>
</tr>
<tr>
<td>Mark nodes as density points</td>
<td>Used to modify the density around a selected vertex.</td>
</tr>
<tr>
<td></td>
<td>Once selected, a method of vertex selection must be activated from the status bar.</td>
</tr>
<tr>
<td>Delete density points</td>
<td>Used to remove any density settings on nodes and remove density points on lines or inside entirely.</td>
</tr>
<tr>
<td>Create density points inside</td>
<td>Used to define density points inside the external contour. Density Points are displayed with a red dot.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If the Snap to Grid option is active, then the density point will be added to the nearest grid intersection.</td>
</tr>
<tr>
<td>Create density regions</td>
<td>Used to define areas within the external contour of higher density.</td>
</tr>
<tr>
<td>Delete density regions</td>
<td>Removes any Density Regions that have been created.</td>
</tr>
<tr>
<td>Border density lines</td>
<td>Used to specify the density setting along an external or internal contour.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Border Density Lines cannot be applied to lines that have had Density Points on Lines applied or already have had a Border Density setting applied. These should be removed and the new Border Density setting applied.</td>
</tr>
<tr>
<td>Internal density lines</td>
<td>Used to specify density lines within the envelope defined by the external contour. Density lines are displayed in green.</td>
</tr>
<tr>
<td>Crack lines</td>
<td>Used to add crack lines to the surface model.</td>
</tr>
<tr>
<td>Delete density lines or crack lines</td>
<td>Used to delete a selected density line or crack line.</td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
</tr>
<tr>
<td>----------------------------------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Create triangulation points on the lines</td>
<td>Used to add triangulation points along the external or polygonal internal contour lines. The Advanced Mesher will be forced to create mesh nodal points in these locations. Triangulation points are displayed with a blue dot.</td>
</tr>
<tr>
<td>Delete triangulation nodes</td>
<td>Removes any density settings on nodes and removes density points on lines or inside entirely. Once selected, a method of node selection must be activated from the status bar.</td>
</tr>
<tr>
<td>Table</td>
<td>Opens the coordinate and density table, which displays data for each of the points that have been defined in the surface model. Both the coordinates and density values can be modified.</td>
</tr>
<tr>
<td><strong>Note:</strong> Additional density and triangulation point coordinates must be modified in such a way as they still remain upon a straight line between the vertices that define the line.</td>
<td></td>
</tr>
<tr>
<td>Density parameters (lines, regions)</td>
<td>Sets or clears the density parameter modification mode. When set, selecting a density line or region opens the density dialog for that line or region allowing you to edit the density value.</td>
</tr>
<tr>
<td>Triangulation</td>
<td>Opens the Triangulation dialog, which is used to specify the meshing method and associated parameters for meshing.</td>
</tr>
<tr>
<td>Change triangulation density ...</td>
<td>Used to select a region in which the mesh density is locally revised after the mesh has been created by defining a density region using either a rectangular or polygonal region.</td>
</tr>
<tr>
<td>Smoothing ...</td>
<td>Opens the Smoothing dialog, which is used to improve the mesh quality to be improved by selecting a criteria and number of times that criteria is repeated.</td>
</tr>
<tr>
<td>Display mesh quality ...</td>
<td>Opens the Mesh Quality dialog, which is used to visualize an aspect of the mesh by color coding to evaluate the mesh quality.</td>
</tr>
<tr>
<td>Mesh quality table ...</td>
<td>Opens the Quality Table, which displays the range of values for a number of parameters for a triangular mesh.</td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Create density points for contours</td>
<td>Used to add triangulation points on external or internal contour lines. As with the vertices for the contours, nodes for the triangulation will always be generated on these points. Density Points are displayed with a red dot.</td>
</tr>
<tr>
<td><strong>Note:</strong> If a vertex at either end of the line is moved, then the density point will be moved in such a way so that the point will remain in the same proportion along the line defined by the new vertices.</td>
<td></td>
</tr>
<tr>
<td>Mark nodes as density points</td>
<td>Used to modify the density around a selected vertex. Once selected, a method of vertex selection must be activated from the status bar.</td>
</tr>
<tr>
<td>Delete density points</td>
<td>Used to remove any density settings on nodes and remove density points on lines or inside entirely.</td>
</tr>
<tr>
<td>Create density points inside</td>
<td>Used to define density points inside the external contour. Density Points are displayed with a red dot.</td>
</tr>
<tr>
<td><strong>Note:</strong> If the Snap to Grid option is active, then the density point will be added to the nearest grid intersection.</td>
<td></td>
</tr>
<tr>
<td>Create density regions</td>
<td>Used to define areas within the external contour of higher density.</td>
</tr>
<tr>
<td>Delete density regions</td>
<td>Removes any Density Regions that have been created.</td>
</tr>
<tr>
<td>Border density lines</td>
<td>Used to specify the density setting along an external or internal contour.</td>
</tr>
<tr>
<td><strong>Note:</strong> Border Density Lines cannot be applied to lines that have had Density Points on Lines applied or already have had a Border Density setting applied. These should be removed and the new Border Density setting applied.</td>
<td></td>
</tr>
<tr>
<td>Internal density lines</td>
<td>Used to specify density lines within the envelope defined by the external contour. Density lines are displayed in green.</td>
</tr>
<tr>
<td>Crack lines</td>
<td>Used to add crack lines to the surface model.</td>
</tr>
<tr>
<td>Delete density lines or crack lines</td>
<td>Used to delete a selected density line or crack line.</td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
</tr>
<tr>
<td>------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Create triangulation points on the lines</td>
<td>Used to add triangulation points along the external or polygonal internal contour lines. The Advanced Mesher will be forced to create mesh nodal points in these locations. Triangulation points are displayed with a blue dot.</td>
</tr>
<tr>
<td>Delete triangulation nodes</td>
<td>Removes any density settings on nodes and removes density points on lines or inside entirely. Once selected, a method of node selection must be activated from the status bar.</td>
</tr>
<tr>
<td>Table</td>
<td>Opens the coordinate and density table, which displays data for each of the points that have been defined in the surface model. Both the coordinates and density values can be modified.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Additional density and triangulation point coordinates must be modified in such a way as they still remain upon a straight line between the vertices that define the line.</td>
</tr>
<tr>
<td>Density parameters (lines, regions)</td>
<td>Sets or clears the density parameter modification mode. When set, selecting a density line or region opens the density dialog for that line or region allowing you to edit the density value.</td>
</tr>
<tr>
<td>Triangulation</td>
<td>Opens the Triangulation dialog, which is used to specify the meshing method and associated parameters for meshing.</td>
</tr>
<tr>
<td>Change triangulation density ...</td>
<td>Used to select a region in which the mesh density is locally revised after the mesh has been created by defining a density region using either a rectangular or polygonal region.</td>
</tr>
<tr>
<td>Smoothing ...</td>
<td>Opens the Smoothing dialog, which is used to improve the mesh quality to be improved by selecting a criteria and number of times that criteria is repeated.</td>
</tr>
<tr>
<td>Display mesh quality ...</td>
<td>Opens the Mesh Quality dialog, which is used to visualize an aspect of the mesh by color coding to evaluate the mesh quality.</td>
</tr>
<tr>
<td>Mesh quality table ...</td>
<td>Opens the Quality Table, which displays the range of values for a number of parameters for a triangular mesh.</td>
</tr>
</tbody>
</table>
### Table 27: Additional edit menu items in the Surface mode

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface by formula</td>
<td>Opens the Formula dialog, which is used to describe the vertical (y-axis) coordinate on any point by a formula.</td>
</tr>
<tr>
<td>Rotate X+</td>
<td>Rotates the view by one rotation angle increment in a positive direction about the global X-axis.</td>
</tr>
<tr>
<td>Rotate X-</td>
<td>Rotates the view by one rotation angle increment in a negative direction about the global X-axis.</td>
</tr>
<tr>
<td>Rotate Y+</td>
<td>Rotates the view by one rotation angle increment in a positive direction about the global Y-axis.</td>
</tr>
<tr>
<td>Rotate Y-</td>
<td>Rotates the view by one rotation angle increment in a negative direction about the global Y-axis.</td>
</tr>
<tr>
<td>Rotate Z+</td>
<td>Rotates the view by one rotation angle increment in a positive direction about the global Z-axis.</td>
</tr>
<tr>
<td>Rotate Z-</td>
<td>Rotates the view by one rotation angle increment in a negative direction about the global Z-axis.</td>
</tr>
<tr>
<td>Rotation Angle</td>
<td>Opens the Rotation Angle dialog, which is used to specify the increment by which the structure is rotated when any of the rotation tools are selected.</td>
</tr>
<tr>
<td>Projection Upon the XZ Plane</td>
<td>Rotates the view of the surface model to display it in the X-Z plane (i.e., the plan view).</td>
</tr>
<tr>
<td>Projection Upon the ZY Plane</td>
<td>Rotates the view of the surface model to display it in the Y-Z plane (i.e., the side elevation).</td>
</tr>
<tr>
<td>Projection Upon the XY Plane</td>
<td>Rotates the view of the surface model to display it in the X-Y plane (i.e., the front elevation).</td>
</tr>
<tr>
<td>Export</td>
<td>Opens the Export dialog, which is used to specify a file name an location for a STAAD.Pro Input file (file extension .STD) so the current surface mesh can be analyzed using STAAD.Pro.</td>
</tr>
</tbody>
</table>

**M. Triangulation dialog**

Used to specify the meshing method and associated parameters for meshing.

Opens when Edit > Triangulation is selected.

Dialog controls

**Mesh Type**

If methods B) or C) are selected, the option to limit the element size become available. This determines the size of the longest edge of the elements that are to be generated.
Element Size
Specify a target element size in the current units.

Element Type
There are four element types:
· Triangular
· Quads from Merged Triangles
· Quads (from Breaking of Triangles)
· Parallel Grid Quads.

Quads from Merged Triangles
Quads (from Breaking of Triangles)

Additional Boundary Nodes
Instructs the program to use

Cancel
Closes the dialog without generating a mesh for the surface.

OK
Closes the dialog and generates a mesh with the selected parameters for the surface.

M. Smoothing dialog
Used to improve the mesh quality by selecting a criteria and number of times that criteria is repeated.

Opens when Edit > Smoothing is selected.

Dialog controls
Smoothing criteria

Number of iteration
Specify a number of times the criteria is to be repeated.

OK
Adds the selected smoothing criteria and closes the dialog.

Cancel
Closes the dialog without adding any smoothing criteria.

M. Mesh Quality dialog
Used to visualize an aspect of the mesh by color coding to evaluate the mesh quality.

Opens when Edit > Mesh Quality is selected.

Dialog controls
The options that are available are:-
· Aspect Ratio Gamma
· Element Area
· Minimum Angle
· Maximum Angle
· Scaled Jacobean
· Ratio of Maximum to Minimum Angles
· Ratio of Maximum to Minimum Edge Length
· Ratio of Triangle Area to the Maximum Area of Connected Triangles
· Ratio of Triangle Area to the Minimum Area of Connected Triangles
· Ratio of the Area to the Square of the Perimeter
· Skew
· Taper
· Stretch
· Oddy
· Jacobean
· Warpage

Note that some of these options are available for triangular or quadrilateral elements only.

The selected item is then divided into a range of between 2 and 16 bands from the minimum to the maximum value. The number and default boundaries can be modified by the user if required.

These defined boundaries are then shown in a table which are colour coded with one of the following colour scales:-

From pale yellow to red
From blue to red
From light grey to black
Multicoloured spectrum

It is additionally possible to select only a range of values to display by having a tick against only those range values that are required. Click on the Turn OFF button to turn all the values off and click on those that are required. Click on the Turn ON button to turn on all the values and click off those values that are not required.

To view the values of the chosen parameter on the elements, click on the Digitize icon on the toolbar.

OK Closes the dialog and opens the Triangle Quality dialog with the range and interval options for the selected mesh criteria. Click OK to display the results graphically on the surface.

Cancel Closes the dialog without selecting a mesh quality criteria.

M. Settings menu
Contains items for controlling non-element specific settings as well as the behavior and display of the drawing environment.

Table 28: Settings menu items

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Preferences ...</td>
<td>Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.</td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Grid Options ...</td>
<td>Opens the Grid Parameters dialog, which is used to specify the grid spacing along the Y and Z directions and the Angle of rotation of the grid axes.</td>
</tr>
<tr>
<td>Grid</td>
<td>Used to set or clear the display of the drawing grid in the View window.</td>
</tr>
<tr>
<td>Coordinate Axis</td>
<td>Used to set or clear the display of the coordinate axis.</td>
</tr>
<tr>
<td>Snap to grid</td>
<td>Used to set or clear the snap to grid behavior, which provides drawing assistance by selecting the nearest grid point to a selected point in the View window.</td>
</tr>
<tr>
<td>Zoom In</td>
<td>Used to increase the magnification in the View window.</td>
</tr>
<tr>
<td>Zoom Out</td>
<td>Used to decrease the magnification in the View window.</td>
</tr>
<tr>
<td>Zoom to Region</td>
<td>Maginifies a windowed area of the View window.</td>
</tr>
<tr>
<td>Show All</td>
<td>Decreases the magnification to display the extents of the overall dimensions.</td>
</tr>
</tbody>
</table>

Table 29: Additional settings menu items in the Surface mode

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Projection</td>
<td>This divides the View window into four views, with each view displaying a different projection of the surface model. The lower right hand corner remains the active view for using the view controls, where the top left, top right, and bottom left views become the Plan, Side Elevation, and Front Elevation views, respectively.</td>
</tr>
</tbody>
</table>

M. Settings dialog
Used to specify unit and output parameters.

Measure units tab
Coordinates included in an exported STAAD.Pro input file use the accuracy specified here.

Other tab

M. Service menu
Contains items for opening tools and utilities related to the Advanced Mesher module.
Table 30: Service menu items

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows Calculator ...</td>
<td>Opens the Windows system Calculator window.</td>
</tr>
</tbody>
</table>

M. Help menu
The Help menu contains items for using online help.

Table 31: Help menu items

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Help Topics</td>
<td>Opens the Advanced Surface Mesher help window.</td>
<td>F1</td>
</tr>
<tr>
<td>About</td>
<td>Opens the About Advanced Mesher window, which displays version and copyright information.</td>
<td></td>
</tr>
</tbody>
</table>

M. Toolbars
The toolbar docked at the top of the window contains commonly used tools for the currently selected mode.

M. Model toolbar
Contains several of the most commonly used tools found in the Advanced Surface Mesher module Model mode.

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Same effect as selecting ...</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Opens an empty surface design file. If the currently loaded section has been modified, then the option is given to save it prior to commencing with the new design.</td>
<td>File &gt; New</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>--------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------------------------------------------------</td>
</tr>
<tr>
<td>![Open Tool Icon]</td>
<td>Opens the Load Model dialog, which is used to select an existing Surface Mesh data file (file extension .SMH). Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the <strong>Preview</strong> option is checked, then a preview of the surface can be displayed prior to opening the file. <strong>Note:</strong> Only a single surface mesh data file may be open at a time, so you will be prompted to save any changes made to the current file, if one is open, and it will be closed prior to opening a different file.</td>
<td>File &gt; Open …</td>
</tr>
<tr>
<td>![Save Tool Icon]</td>
<td>Saves any changes made to the current data file. <strong>Note:</strong> If the current file has not been previously saved, this has the same effect as selecting <strong>Save As</strong>…</td>
<td>File &gt; Save</td>
</tr>
<tr>
<td>![Undo Tool Icon]</td>
<td>Negates the last drawing operation. You can undo a series of modifications by repeatedly selecting <strong>Undo</strong>.</td>
<td>Edit &gt; Undo</td>
</tr>
<tr>
<td>![Redo Tool Icon]</td>
<td>Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting <strong>Redo</strong>.</td>
<td>Edit &gt; Redo</td>
</tr>
<tr>
<td>![Cancel Operation Tool Icon]</td>
<td>Cancels the current drawing operation.</td>
<td>ESC</td>
</tr>
<tr>
<td>![Overall Dimensions Tool Icon]</td>
<td>Opens the Overall Dimensions dialog, which is used to specify the size of the workspace that will be required to create the section. The minimum it should be set to is the overall dimensions of the section but can be set larger.</td>
<td>Edit &gt; Overall Dimensions …</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>----------------------</td>
<td>------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------------------------------------------------</td>
</tr>
<tr>
<td><img src="image" alt="Grid Setup" /></td>
<td>Opens the Grid Parameters dialog, which is used to specify the grid spacing along the Y and Z directions and the Angle of rotation of the grid axes.</td>
<td>Settings &gt; Grid Options …</td>
</tr>
<tr>
<td><img src="image" alt="External Contour" /></td>
<td>Used to set or clear the external contour drawing mode, which defines the outside of the section.</td>
<td>Edit &gt; Draw External Contour</td>
</tr>
<tr>
<td><img src="image" alt="Internal Contour" /></td>
<td>Used to set or clear the internal contour mode, which is used for creating internal openings within an external boundary.</td>
<td>Edit &gt; Draw Internal Contour</td>
</tr>
<tr>
<td><img src="image" alt="Circle hole" /></td>
<td>Used to set or clear the circular hole drawing mode, which is used to define the center of a hole (e.g., a parametrically defined, circular inner contour) and radius graphically.</td>
<td>Edit &gt; Insert circular hole, center and radius</td>
</tr>
<tr>
<td><img src="image" alt="Circle hole with specified radius" /></td>
<td>Insert circular hole with defined radius</td>
<td>Edit &gt; Insert circular hole with defined radius</td>
</tr>
<tr>
<td><img src="image" alt="Smooth" /></td>
<td>Used to set or clear the smooth corner drawing mode, which is used to select a corner to fillet and then opens the Smooth radius dialog to specify a radius value for the rounded corner.</td>
<td>Edit &gt; Radius Corner…</td>
</tr>
<tr>
<td><img src="image" alt="Delete" /></td>
<td>Used to set or clear the delete internal contour drawing mode, which is used to remove internal contours by clicking anywhere in their boundary.</td>
<td>Edit &gt; Delete internal contour</td>
</tr>
<tr>
<td><img src="image" alt="Transformation" /></td>
<td>Used to set or clear the delete internal contour drawing mode, which is used to remove internal contours by clicking anywhere in their boundary.</td>
<td>Edit &gt; Transformation</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
<td>-----------------------------</td>
</tr>
<tr>
<td><img src="image1" alt="Tool" /></td>
<td><strong>Move Vertices</strong></td>
<td>Used to window either a rectangular or polygonal area to select one or more vertices to move. Vertices can be moved either manually or by specifying offset distances.</td>
</tr>
<tr>
<td><img src="image2" alt="Tool" /></td>
<td><strong>Copy inner contour(s)</strong></td>
<td>Used to window either a rectangle or polygonal area to select one or more internal contours for copying. Copies can be placed manually or by specifying offset distances.</td>
</tr>
<tr>
<td><img src="image3" alt="Tool" /></td>
<td><strong>Multiple copies of inner contour(s)</strong></td>
<td>Used to window either a rectangle or polygonal area to select one or more internal contours making multiple copying. Copies can be placed manually or by specifying offset distances and a number of copies.</td>
</tr>
<tr>
<td><img src="image4" alt="Tool" /></td>
<td><strong>Delete Vertices</strong></td>
<td>Used to select a single vertex or window either a rectangular or polygonal area to select one or more vertices for deletion.</td>
</tr>
<tr>
<td><img src="image5" alt="Tool" /></td>
<td><strong>Vertices</strong></td>
<td>Displays all vertex numbers in the View window and opens the Coordinates dialog which is used to specify precise coordinates of each vertex for a selected contour.</td>
</tr>
<tr>
<td><img src="image6" alt="Tool" /></td>
<td><strong>Preferences</strong></td>
<td>Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.</td>
</tr>
<tr>
<td><img src="image7" alt="Tool" /></td>
<td><strong>Grid</strong></td>
<td>Used to set or clear the display of the drawing grid in the View window.</td>
</tr>
<tr>
<td><img src="image8" alt="Tool" /></td>
<td><strong>Coordinate Axis</strong></td>
<td>Used to set or clear the display of the coordinate axis.</td>
</tr>
</tbody>
</table>
### Tool Name

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Same effect as selecting …</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="Image" alt="Snap to Grid" /></td>
<td>Used to set or clear the snap to grid behavior, which provides drawing assistance by selecting the nearest grid point to a selected point in the View window.</td>
<td><strong>Settings &gt; Snap to Grid</strong></td>
</tr>
<tr>
<td><img src="Image" alt="Zoom In" /></td>
<td>Used to increase the magnification in the View window.</td>
<td><strong>Settings &gt; Zoom In</strong></td>
</tr>
<tr>
<td><img src="Image" alt="Zoom Out" /></td>
<td>Used to decrease the magnification in the View window.</td>
<td><strong>Settings &gt; Zoom Out</strong></td>
</tr>
<tr>
<td><img src="Image" alt="Zoom to Region" /></td>
<td>Magnifies a windowed area of the View window.</td>
<td><strong>Settings &gt; Zoom to Region</strong></td>
</tr>
<tr>
<td><img src="Image" alt="Show All" /></td>
<td>Decreases the magnification to display the extents of the overall dimensions.</td>
<td><strong>Settings &gt; Show All</strong></td>
</tr>
<tr>
<td><img src="Image" alt="Help" /></td>
<td>Opens the Advanced Surface Mesher help window.</td>
<td><strong>Help &gt; Help Topics</strong></td>
</tr>
<tr>
<td><img src="Image" alt="About" /></td>
<td>Opens the About Advanced Mesher window, which displays version and copyright information.</td>
<td><strong>Help &gt; About ...</strong></td>
</tr>
</tbody>
</table>

### M. Triangulation toolbar
Contains several of the most commonly used tools found in the Advanced Surface Mesher module Triangulation mode.
<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Same effect as selecting …</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Tool" /></td>
<td><strong>Create density points on the contours</strong></td>
<td>Used to add triangulation points on external or internal contour lines. As with the vertices for the contours, nodes for the triangulation will always be generated on these points. Density Points are displayed with a red dot. <strong>Note:</strong> If a vertex at either end of the line is moved, then the density point will be moved in such a way so that the point will remain in the same proportion along the line defined by the new vertices</td>
</tr>
<tr>
<td><img src="image2" alt="Tool" /></td>
<td><strong>Mark nodes as density points</strong></td>
<td>Used to modify the density around a selected vertex. Once selected, a method of vertex selection must be activated from the status bar.</td>
</tr>
<tr>
<td><img src="image3" alt="Tool" /></td>
<td><strong>Create density points inside</strong></td>
<td>Used to define density points inside the external contour. Density Points are displayed with a red dot. <strong>Note:</strong> If the Snap to Grid option is active, then the density point will be added to the nearest grid intersection.</td>
</tr>
<tr>
<td><img src="image4" alt="Tool" /></td>
<td><strong>Delete density points</strong></td>
<td>Used to remove any density settings on nodes and remove density points on lines or inside entirely.</td>
</tr>
<tr>
<td><img src="image5" alt="Tool" /></td>
<td><strong>Internal density lines</strong></td>
<td>Used to specify density lines within the envelope defined by the external contour. Density lines are displayed in green.</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td><img src="image" alt="Border density lines" /></td>
<td>Used to specify the density setting along an external or internal contour. <strong>Note:</strong> Border Density Lines cannot be applied to lines that have had Density Points on Lines applied or already have had a Border Density setting applied. These should be removed and the new Border Density setting applied.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Crack lines" /></td>
<td>Used to add crack lines to the surface model.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Delete density lines or crack lines" /></td>
<td>Used to delete a selected density line or crack line.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Create density region" /></td>
<td>Used to define areas within the external contour of higher density.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Delete density region" /></td>
<td>Removes any Density Regions that have been created.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Density parameter (lines, regions)" /></td>
<td>Sets or clears the density parameter modification mode. When set, selecting a density line or region opens the density dialog for that line or region allowing you to edit the density value.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Create triangulation points on the lines" /></td>
<td>Used to add triangulation points along the external or polygonal internal contour lines. The Advanced Mesher will be forced to create mesh nodal points in these locations. Triangulation points are displayed with a blue dot.</td>
<td></td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td><strong>Delete triangulation nodes</strong>&lt;br&gt;Removes any density settings on nodes and removes density points on lines or inside entirely.&lt;br&gt;Once selected, a method of node selection must be activated from the status bar.</td>
<td></td>
</tr>
<tr>
<td><img src="image2.png" alt="Image" /></td>
<td><strong>Table</strong>&lt;br&gt;Opens the coordinate and density table, which displays data for each of the points that have been defined in the surface model. Both the coordinates and density values can be modified.  &lt;br&gt;&lt;br&gt;&lt;b&gt;Note:&lt;/b&gt; Additional density and triangulation point coordinates must be modified in such a way as they still remain upon a straight line between the vertices that define the line.</td>
<td></td>
</tr>
<tr>
<td><img src="image3.png" alt="Image" /></td>
<td><strong>Triangulation</strong>&lt;br&gt;Opens the Triangulation dialog, which is used to specify the meshing method and associated parameters for meshing.</td>
<td>Edit &gt; Triangulation</td>
</tr>
<tr>
<td><img src="image4.png" alt="Image" /></td>
<td><strong>Change triangulation density</strong>&lt;br&gt;Used to select a region in which the mesh density is locally revised after the mesh has been created by defining a density region using either a rectangular or polygonal region.</td>
<td></td>
</tr>
<tr>
<td><img src="image5.png" alt="Image" /></td>
<td><strong>Smoothing</strong>&lt;br&gt;Opens the Smoothing dialog, which is used to improve the mesh quality to be improved by selecting a criteria and number of times that criteria is repeated.</td>
<td>Edit &gt; Display mesh quality</td>
</tr>
<tr>
<td><img src="image6.png" alt="Image" /></td>
<td><strong>Display mesh quality</strong>&lt;br&gt;Opens the Mesh Quality dialog, which is used to visualize an aspect of the mesh by color coding to evaluate the mesh quality.</td>
<td></td>
</tr>
<tr>
<td><img src="image7.png" alt="Image" /></td>
<td><strong>Digitize</strong>&lt;br&gt;Sets or clears the display of values for the Mesh Quality on the surface in the View window.</td>
<td></td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting ...</td>
</tr>
<tr>
<td>----------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------------------------------</td>
</tr>
<tr>
<td>Mesh quality table</td>
<td>Opens the Quality Table, which displays the range of values for a number of parameters for a triangular mesh.</td>
<td>Edit &gt; Mesh Quality Table ...</td>
</tr>
<tr>
<td>Preferences</td>
<td>Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.</td>
<td>Settings &gt; Preferences ...</td>
</tr>
<tr>
<td>Grid</td>
<td>Used to set or clear the display of the drawing grid in the View window.</td>
<td>Settings &gt; Grid</td>
</tr>
<tr>
<td>Coordinate Axis</td>
<td>Used to set or clear the display of the coordinate axis.</td>
<td>Settings &gt; Coordinate Axis</td>
</tr>
<tr>
<td>Snap to Grid</td>
<td>Used to set or clear the snap to grid behavior, which provides drawing assistance by selecting the nearest grid point to a selected point in the View window.</td>
<td>Settings &gt; Snap to Grid</td>
</tr>
<tr>
<td>Zoom In</td>
<td>Used to increase the magnification in the View window.</td>
<td>Settings &gt; Zoom In</td>
</tr>
<tr>
<td>Zoom Out</td>
<td>Used to decrease the magnification in the View window.</td>
<td>Settings &gt; Zoom Out</td>
</tr>
<tr>
<td>Zoom to Region</td>
<td>Magnifies a windowed area of the View window.</td>
<td>Settings &gt; Zoom to Region</td>
</tr>
<tr>
<td>Show All</td>
<td>Decreases the magnification to display the extents of the overall dimensions.</td>
<td>Settings &gt; Show All</td>
</tr>
<tr>
<td>Help</td>
<td>Opens the Advanced Surface Mesher help window.</td>
<td>Help &gt; Help Topics</td>
</tr>
</tbody>
</table>
### M. Surface toolbar

Contains several of the most commonly used tools found in the Advanced Surface Mesher module Surface mode.

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Same effect as selecting ...</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Help Icon" /></td>
<td>About</td>
<td>Help &gt; About ...</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Surface by formula</td>
<td>Edit &gt; Surface by formula</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Export</td>
<td>Edit &gt; Export ...</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotation Angle</td>
<td>Edit &gt; Rotation Angle ...</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotate X-</td>
<td>Edit &gt; Rotate X-</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotate X+</td>
<td>Edit &gt; Rotate X+</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotate Y-</td>
<td>Edit &gt; Rotate Y-</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>-----------------------------</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotate Y+</td>
<td>Edit &gt; Rotate Y+</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotates the view by one rotation angle increment in a positive direction about the global Y-axis.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotate Z-</td>
<td>Edit &gt; Rotate Z-</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotates the view by one rotation angle increment in a negative direction about the global Z-axis.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotate Z+</td>
<td>Edit &gt; Rotate Z+</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotates the view by one rotation angle increment in a positive direction about the global Z-axis.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Projection upon the XZ plane</td>
<td>Edit &gt; Projection upon the XZ plane</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotates the view of the surface model to display it in the X-Z plane (i.e., the plan view).</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Projection upon the XY plane</td>
<td>Edit &gt; Projection upon the XY plane</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotates the view of the surface model to display it in the X-Y plane (i.e., the front elevation).</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Projection upon the ZY plane</td>
<td>Edit &gt; Projection upon the ZY plane</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Rotates the view of the surface model to display it in the Y-Z plane (i.e., the side elevation).</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>Show Projections</td>
<td>Settings &gt; Show Projections</td>
</tr>
<tr>
<td><img src="image" alt="Tool Icon" /></td>
<td>This divides the View window into four views, with each view displaying a different projection of the surface model. The lower right hand corner remains the active view for using the view controls, where the top left, top right, and bottom left views become the Plan, Side Elevation, and Front Elevation views, respectively.</td>
<td></td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>------------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>-----------------------------</td>
</tr>
</tbody>
</table>
| ![Table icon]               | Opens the coordinate and density table, which displays data for each of the points that have been defined in the surface model. Both the coordinates and density values can be modified.  
  **Note:** Additional density and triangulation point coordinates must be modified in such a way as they still remain upon a straight line between the vertices that define the line. | Edit > Table                |
| ![Change triangulation density icon] | Used to select a region in which the mesh density is locally revised after the mesh has been created by defining a density region using either a rectangular or polygonal region. | Edit > Change triangulation density |
| ![Display mesh quality icon] | Opens the Mesh Quality dialog, which is used to visualize an aspect of the mesh by color coding to evaluate the mesh quality.                                                                                     | Edit > Display mesh quality  |
| ![Mesh quality table icon]  | Opens the Quality Table, which displays the range of values for a number of parameters for a triangular mesh.                                                                                                  | Edit > Mesh Quality Table ... |
| ![Preferences icon]         | Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.                                                                                         | Settings > Preferences ...   |
| ![Zoom In icon]             | Used to increase the magnification in the View window.                                                                                                                                                       | Settings > Zoom In          |
| ![Zoom Out icon]            | Used to decrease the magnification in the View window.                                                                                                                                                       | Settings > Zoom Out         |
### Tool Description

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Same effect as selecting …</th>
</tr>
</thead>
<tbody>
<tr>
<td>⚫️</td>
<td>Decreases the magnification to display the extents of the overall dimensions.</td>
<td>Settings &gt; Show All</td>
</tr>
<tr>
<td>Show All</td>
<td></td>
<td></td>
</tr>
<tr>
<td>📕</td>
<td>Opens the Advanced Surface Mesher help window.</td>
<td>Help &gt; Help Topics</td>
</tr>
<tr>
<td>Help</td>
<td></td>
<td></td>
</tr>
<tr>
<td>🎨</td>
<td>Opens the About Advanced Mesher window, which displays version and copyright information.</td>
<td>Help &gt; About …</td>
</tr>
<tr>
<td>About</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### M. Composite Decks

STAAD.Pro can model composite steel deck systems for design.

An important feature to note is that when the structure is modified, the deck automatically gets modified with it. Note that when the dimension of the structure is changed along the length from 30 ft to 40 ft, the deck gets modified with the structure and there is no need to redefine the deck all over again. The deck loading is also revised automatically.

![Figure 106: Composite deck and load update with structure changes](image)

The design parameters for the composite deck are already defined through the interface. Few additional design parameters like TRACK can be assigned, if so desired, and finally, design commands like CHECK CODE has to be assigned to the composite deck.

### Related Links

- [Composite Deck dialog](on page 2916)
- [TR.20.7 Composite Decks](on page 2473)
- [M. To add a floor load or one-way load](on page 831)
M. To create a new composite deck from perimeter beams

1. Select the **Composite Deck Layout** tool in the **Composite Deck** group on the **Geometry** ribbon tab.

   The **Composite Deck** dialog opens.

2. Select the perimeter beams:
   a. Select the **Clicking on Nodes** option.
   b. Click **Create New Deck**.

   The mouse pointer changes to the Add Composite Deck cursor ( ).
   c. Click on the end nodes defining the corners of the perimeter beam area.
   d. Click on the first node a second time to close the decking area.

   Alternatively, you can select the perimeter beams in the view window first. Then select the **Use Selected Beam** option and click **Create New Deck**.

   The **New Composite Deck** dialog opens.

3. Type a **New Composite Deck Name** and then click **OK**.

   The deck name is added to the list of composite decks with all the perimeter and interior beams listed.

   **Note:** You can delete a composite deck definition by selecting it in within the list in the **Composite Deck** dialog and then pressing **<Delete>**. Confirm you want to delete the deck in the dialog. Only the deck definition is deleted. Constituent members will remain in the model.

**Related Links**
- **Composite Deck dialog** (on page 2916)

M. To specify a direction for the composite deck ribs

To specify the orientation of the deck ribs, use the following procedure.

The program will automatically assign a rib direction parallel to the detected shortest span in the deck area. However, you will often need to edit the orientation.

1. Select a composite deck name in the **Composite Deck** dialog list.
2. Select a pair of parallel beams.

   The deck ribs will run perpendicular to these two beams.

3. Click **Create Direction**.

   If you have previously assigned a deck rib direction, you will be prompted to confirm you want to update this direction.

   The deck span direction arrow is updated to reflect the change in direction.

**Related Links**
- **Composite Deck dialog** (on page 2916)
M. To assign composite deck properties

To define the concrete, rib, and connectivity details of a composite deck, use the following procedure.

1. Select a composite deck name in the Composite Deck dialog list.
2. Type values to use for the concrete in the current units:
   - Concrete thickness above flutes
   - Unit Wt. of Concrete
   - Concrete Grade
3. Specify the steel deck rib properties:
   - To...                        Do the following...
     use a catalog value          select the deck name from the Use Database drop-down list
     specify a custom value       type values for Rib Width and Rib Height.

   Note: The program includes three standard catalogs: ASC™, Vulcraft™, and VERCO™.

4. Specify the Stud Diameter by either:
   - selecting a predefined diameter from the drop-down list
   - or selecting Custom in the drop-down list and then typing the diameter in the current units
5. Type the Stud Length in the current units.
6. Select if the composite deck is Shored (i.e., propped) or Unshored during construction.
7. Click Update Deck Property.

Related Links
- Composite Deck dialog (on page 2916)

M. To modify composite steel beam properties

To change the steel beam profile or effective flange width, use the following procedure.

The program will use the assigned steel sections for beams and also automatically calculate the effective flange width of the composite section. However, you can update steel sections from the Composite Decks dialog as well as specify a value for the effective flange width.

1. Select a beam in the Composite Decks dialog within a composite deck name.
   The beam options are displayed in the dialog.
2. (Optional) Click Add/Change Property to open the Section Profile Tables dialog.
   From there, you can select a different beam or beam properties.

   Note: You should not specify a composite section type specification for these beams. The use of the composite deck definition allows the program to calculate all the necessary composite section parameters automatically. Refer to G.6.7 Composite Beams and Composite Decks (on page 2328) for details.
3. (Optional) Type an Effective Width in the current units and then click Update to change the effective width of the composite deck for the beam.
M. Solids

M. To draw a solid connecting existing nodes

To draw a solid with 4 to 8 nodes connecting existing nodes, use the following procedure. You model must have at least four nodes which do not all lie in the same plane. The basic solid element consists of 8 nodes. However, by collapsing nodes together progressively, different three-dimensional solid shapes can be formed. The minimal number of nodes is a four-noded, pyramidal shape.

1. On the Geometry ribbon tab, select one of the tools in the Add Solid menu in the Solid group.

   Use this tool...  To...
   
   create eight-noded solid elements (i.e., a cuboid)

   create seven-noded solid elements

   create six-noded solid elements (i.e, a triangular prism)

   create five-noded solid elements (i.e., a rectangular pyramid)

   create four-noded solid elements (i.e., a triangular pyramid, or tetrahedron)

   The mouse pointer changes to either the Add Solid cursor ( ).

2. Click on any node that will form the first corner of the solid element. A line is “rubber banded” to the cursor from this node. This represents the first edge of the solid.

3. Continue clicking on nodes up to the number corresponding to the tool selected in step 1. The solid element is added to the model.

Related Links
M. Nodes

To insert a node in a single member

Used to insert nodes in an existing member(s). The member is split into the corresponding number of segments with automatic generation of node numbers, member numbers, member properties, and loads.

1. On the Geometry ribbon tab, select the Insert Node tool in the Beam group.

   ![Insert Node tool]

   The mouse pointer changes to the Insert Node pointer.

2. Select a beam member.

   The Insert Nodes into Beam # dialog (on page 2906) opens.

3. Either:

   - Specify a Distance or Proportion value and click Add New Point.
   - Click Add Mid Point to add a new node halfway along the length of the beam.
   - Specify an integer number of new nodes in the \( n = \) box and click Add n Points to divide the beam into \( n+1 \) number of equal divisions.

   The new node point is listed in the Insertion Points list.

4. Repeat Step 3 as many times as needed.

   Tip: Click Remove to remove the selected node point from the Insertion Points list if you make an error.

5. Click OK.

   The dialog closes and the specified nodes are inserted into the beam.

Related Links

- Insert Nodes into Beam # dialog (on page 2906)

To insert a node in multiple members

Used to insert nodes in an existing member(s). The member is split into the corresponding number of segments with automatic generation of node numbers, member numbers, member properties, and loads.

1. Select two or more beam elements.

2. On the Geometry ribbon tab, select the Insert Node tool in the Beam group.
The **Insert Node/Nodes** dialog (on page 2907) opens.

3. Either:
   - Select the **New point by distance** option and then type a **Distance** value.
   - or
   - Select the **New point by proportion** option and then type a **Proportion** value.
   - or
   - Select the **Add Mid Point** option.
   - or
   - Select the **Add 'n' Points** option and then type an integer number of new nodes in the \( n = \) box to divide the beam into \( n + 1 \) number of equal divisions.

4. Click **OK**.
   - The dialog closes and the specified nodes are inserted into the beams.

**Related Links**
- **Insert Node / Nodes dialog** (on page 2907)

**M. To add a node at overlapping beams**

To connect to overlapping beams with a node at that location, use the following procedure.

You can manually select beams that are overlapping or the program can automatically detect these for you using the optional first step below.

1. (Optional) Automatically select overlapping beams:
   - a. On the **Geometry** ribbon tab, select the **Intersecting Beams > Highlight Intersecting Beams** tool in the **Beam** group.

   The **Intersect Members** dialog opens.
   - b. In the **Enter Tolerance** field, type a tolerance used for how close members should pass before the program considers them to be overlapping.
   - c. Click **OK**.
   - The program automatically selects all overlapping beam sets.

2. On the **Beam Tools** ribbon tab, select the **Intersect Beams** tool in the **Model** group.

3. Type a tolerance to use in the **Intersect Members** dialog and then click **OK**.
   - A message dialog opens to indicate that new beams have been created.
M. To merge two nodes

1. Select the nodes to be merged using the Nodes Cursor.
2. On the Geometry ribbon tab, select the Merge Selected Nodes tool in the Node group.

The Select Node dialog opens.
3. Select the Node To Keep from the list of selected nodes.
   The remaining nodes will be removed. All members and elements which were connected to those nodes will
   be reconnected to the selected node to keep.
4. Click OK.

M. Modify Your Model

The following procedures are used to manipulate existing model geometry.

M. To move selected objects

You must have one or more objects selected.
Refer to GS. Selecting Objects in STAAD.Pro (on page 37) for details.

1. On the Geometry ribbon tab, select one of the following tools:
   - the Move Node tool in the Node group
   or
   - the Move Beam tool in the Beam group
   or
   - the Move Plate tool in the Plate group
   or
   - the Move Solid tool in the Solid group
   The Move Entities dialog opens.
Note: This dialog has a different title depending on the object type being moved but the functionality is similar.

2. Select the method by which the objects are to be moved:

   By distance between following nodes uses two node numbers and determines the vector between them. The selected entities are moved along a parallel vector of the same distance.

   or

   By the following X, Y and Z values uses the Cartesian coordinates specified to create a vector.

3. Specify node numbers or vector component values.

4. (Optional) Check the Retain connections option to retain connections between entities.

   Note: The model geometry will be adjusted to some degree, even if this is not selected, when nodes are the entities being moved as they member, plate, surface, and solid geometry is defined by node location. However, if you are moving a member, plate, surface, or solid, then selecting this option will move the associated nodes and thus retain the connection with adjacent entities (though their geometry will stretch, warp, or shift as a result of the move).

5. Click OK.

   A message dialog opens asking if you would like new nodes created where any moved entities cross other existing entities.

6. Click Yes to create new nodes or No to ignore intersections.

   The model is updated with the moved geometry.

Related Links

• Move Entities dialog (on page 2902)

M. To move the model origin

To shift the translate the entire model by shifting the origin location along a vector, use the following procedure.

Tip: You can use the Generate Rotate tool to rotate the entire model about the origin as well.

The Move Origin dialog opens.

2. Specify the origin offset vector method:

<table>
<thead>
<tr>
<th>Vector method</th>
<th>Do the following…</th>
</tr>
</thead>
<tbody>
<tr>
<td>two nodes</td>
<td>select the By distance between following two nodes and then type the node numbers in the corresponding fields.</td>
</tr>
<tr>
<td>global axes components</td>
<td>select the By the following X, Y and Z values and then type the global axes component distances in the corresponding fields.</td>
</tr>
</tbody>
</table>

Either method generates a vector by which the origin is shifted.

**Note:** A positive value in the coordinates (or resulting positive value resulting the from the selected nodes) will move the origin in a positive direction along the corresponding global axis. Thus, the resulting coordinate of each node is therefore reduced by that same magnitude.

For example, providing a origin shift along the X axis of 5 units, means that a node previously located at the origin now has an X coordinate of -5 units.

3. Click OK.

The coordinates of each node are modified in the STAAD input file to reflect the change in distance to the new origin.

**Related Links**
- Move Origin dialog (on page 2903)

**M. To rotate selected entities**

You must have one or more objects selected.

Refer to GS. Selecting Objects in STAAD.Pro (on page 37) for details.

1. On the Geometry ribbon tab, select the Rotate tool in the Structure group.

   The Rotate dialog opens.

2. Specify an Angle of rotation.

3. Provide two points used to define the axis of rotation by either:

   - selecting two Nodes
   - or
   - typing coordinates of two arbitrary Points

4. Select if the selected geometry is to be copied or moved and, if copied, if linking members are to be created.

5. Click OK.

**Related Links**
M. To generate copies of geometry along a line

To generate copies of selected geometry along a path parallel to a global axis, use the following procedure.

1. Select the structure geometry (beams, plates, solids, etc.) that you want to copy.

   **Tip:** You do not need to select the nodes, as they will be copied along with the other model entities.

2. On the **Geometry** ribbon tab, select the **Translational Repeat** tool in the **Structure** group.

   The **Translational Repeat** dialog opens.

3. Select the **Global Direction** along which the copies will be generated.

4. Specify the **No. of Steps** to generate.

   **Tip:** By setting the **Default Spacing** value first, the entries will update with the new value for each step.

5. (Optional) Edit the **Spacing** values for each step (copy) if the steps are irregular or if the desired **Default Spacing** was not first specified.

6. (Optional) Set the option to **Renumber Bay** and specify a **Number from** value (start number of that copy) if needed.

7. (Optional) Set the **Link Steps** option to generate additional members between copies, parallel to the **Global Direction**.

   You may also set the **Open Base** option, which will prevent links from being generated between nodes with the lowest Y coordinates (when Y is vertical/up).

8. (Optional) Set the **Generation Flags** to the level of geometry, properties, or specifications to be copied to the new, generated members.

9. Click **OK**.

   The new members are generated using the specified parameters.

**Related Links**

- **Translational Repeat dialog** (on page 2892)

M. To generate copies of geometry along an arc

To create copies of selected geometry along an arc about a global axis, use the following procedure.

1. Select the structure geometry (beams, plates, solids, etc.) that you want to copy.

   **Tip:** You do not need to select the nodes, as they will be copied along with the other model entities.

2. On the **Geometry** ribbon tab, select the **Circular Repeat** tool in the **Structure** group.

   The **3D Circular** dialog opens.
3. Select the global **Axis of Rotation** about which the copies will be generated.

4. Specify the **Total Angle** of the arc.
   
   This is measured in a positive (right-hand rule) direction about the selected **Axis of Rotation** between the original selection and the last copy.

5. Specify the **No. of Steps** to generate.
   
   The copies will be equally distributed about the specified arc.

6. Specify a center of rotation by either:
   
   - typing a Node number. You may also click the Node Selection tool to graphically select a node.
   - or
   
   planar coordinates for an arbitrary point (i.e., not at a node)

   **Note:** The axis of rotation for the circular copy will pass through this selected node or point, parallel to the selected global axis of rotation.

7. (Optional) Check the **Use this as Reference Point for Beta angle generation** option to orient the copies towards the central axis of rotation.

8. (Optional) Set the **Link Steps** option to generate additional members between copies.
   
   You may also set the **Open Base** option, which will prevent links from being generated between nodes with the lowest Y coordinates (when Y is vertical/up).

9. (Optional) Set the **Geometry Only** option to generate only nodes, members, elements, etc. with no properties assigned.

10. Click **OK**.

   The new members are generated using the specified parameters.

**Related Links**

- **3D Circular dialog** (on page 2893)

**M. To generate mirror copies of model entities**

To generate symmetric copies of the model selection about a plane, use the following procedure.

1. Select the geometry to copy or move.

   **Note:** Refer to the discussion on the Select menu for details on the selection procedure.

2. On the **Geometry** ribbon tab, select the **Mirror** tool in the **Structure** group.

   The **Mirror** dialog opens.

3. Specify an mirror plane.

4. Specify a Plane position.

5. Select if the selected geometry is to be copied of moved and, if copied, if linking members are to be created.

6. (Optional) Set the option to mirror members if their orientation is also to be mirrored.

7. Click **OK**.

**Related Links**
M. Groups

A group is a way to collect a series of objects together for selection, loading, or design purposes.

M. To create a group from a selection

To create a new group using a selection of model objects, use the following procedure.

Group names are a means for easily identifying a collection of entities like Beams, Plates or Solids using a single moniker. By grouping these entities, we need to assign attributes such as member properties and material constants just to the group, a simple process, compared to the task of assigning them to the individual members.

The file EX. US-1 Plane Frame with Steel Design (on page 4313) is used to demonstrate this feature.

Consider the members which form the truss as being in one of three groups: Top Chords, Bottom Chords, and Web Members.

1. Select the top chord members:
   With the model open, use the Beam Cursor tool to select the members forming the top chords (i.e., members 8 through 13).

   Tip: Press and hold the <Ctrl> key and then click the members in the View window.

2. Either:
   On the Utilities ribbon tab, select the Groups tool in the Geometry Tools group.
or
press <Ctrl+G>

If you have not previously defined any groups, the Define Group Name dialog opens. Otherwise, the Create Group dialog opens (skip to step 4).

3. Specify the group details:
   a. Type a name in the Group Name field.
      
      Note: Group Names must begin with the underscore "_" character.
      
      A group name such as _TOPCHORD can be used for this example.
   b. Select the object type from the Select Type drop-down list.
      
      Groups can consist of nodes, beams, plates, solids, and a general category called geometry. Geometry can be used to associated objects of mixed types.
      Select Beam for the top chords example.
   c. Click OK.

      The Create Group dialog opens. At this point, you have created a group name but it does not yet contain any objects.
4. Select **Associate to Selected Geometry** and then click **Associate**.

The selected members are added to the group.

5. Select other model objects.

**Tip:** The **Create Group** dialog can remain open while you change selection tools and select objects in the View window.

For the example file, the members forming the bottom chord (members 20, 21, 22, and 23).
6. Associated these members with a new group:
   a. Click **Create** in the **Create Group** dialog.
   b. Follow steps 4 and 5 and assign the **Group Name** and objects type as the **Select Type**.
      For the example file US-1 Plane Frame with Steel Design.std, use the name _BOTCHORD for the bottom chord members.

7. Repeat steps 5 and 6 to create additional groups as needed.
   For the example file US-1 Plane Frame with Steel Design.std, you can select the truss web members (members 14, 15, 16, 17, 18, and 19) and add them to a group named _TRUSSWEB
8. Click Close in the Create Group dialog.

Select the Group Selection tool in the Selection group on the Selection ribbon tab to open the Select Groups dialog, which can be used to select a named group.
Note: If the option for **Show Label** are unchecked, labels such as beam numbers will not be displayed for these specific members, even if the beam numbering icon is switched on for the entire structure.

**Related Links**
- *Create Group dialog* (on page 3099)
- *TR.16.1 Listing of Entities by Specifying Groups* (on page 2440)

**M. To add objects to an existing group**

To add more objects to an existing group, use the following procedure.

It is important to select the current objects in a group along with the additional objects so that the current objects are not removed. When associating objects to a group, *only* the current selection (or other association method) will be included in the group.

1. On the Utilities ribbon tab, select the Groups tool in the Geometry Tools group.

![Create Groups dialog](image)

   The **Create Groups** dialog opens.

2. Select the group name to which you want to add objects.

3. Click **Highlight**.

   The current objects in the group are selected in the view window.

4. In the view window, use the selection cursors to select the additional geometry.

   **Note**: Hold the `<Ctrl>` key in order to select additional objects without unselecting the current group contents.

5. Select the **Associated to Selected Geometry** option.

6. Click **Associate**.
M. Structure Wizard

Used to parametrically generate a structural model and then transfer and superimpose it on the current structure in STAAD.Pro.

Opens when **Geometry > Run Structure Wizard** is selected in the STAAD.Pro window.

The **Prototype Models** and **Saved User Models** options appear on the top of the left side of the screen. If the **Prototype Models** option is selected, the **Model Type** will list the types of prototype structures available (such as Trusses, Frames, Plates, Solids, etc.) as shown below. If the **Saved User Models** option is selected, the **Model Type** will display the list previously done and saved models by the user.
To set units

To set the units of length used in the parametric model types, use the following procedure.

The current units of length are displayed in the Structure Wizard status bar.

1. Select **File > Select Units**.
   
   The **Select Units** dialog opens

2. Select one of the units of length.
   
   - Inches
   - Feet
   - Millimeters
   - Centimeters
   - Meters

3. Click **OK**

**M. Generation of Structure from Models**

In this section, the process of generating a structural model and combining it with the existing STAAD.Pro structure will be explained using a Howe Roof truss. Follow these steps to create the other truss types also.

**Selection of Units**

The unit of length should be specified prior to the generation of a model. Select **File > Select Unit** and the **Select Unit** dialog opens. You can select any unit of length from Imperial (inch, feet) or SI/Metric (millimeter, centimeter, meter) system of units.
To create a truss model

1. Select **Truss Models** from the **Model Type** drop-down list.
2. Select one of the parametric truss types by either:
   - double-click on the type of model
   - or
   - click-and-drag the model icon into the view window

   - **Pratt Truss**
   - **Warren Truss**
   - **Howe Bridge**
   - **Lattice Truss**
   - **Howe Roof**
   - **North Light**

   The **Select Parameters** dialog opens.

3. (Optional) Type a **Model Name**.
4. Type the overall truss **Length**, **Height**, and **Width** distances.
5. Type the **No. of bays along length** to define the number of equally spaced bay along the length of the truss.
6. (Optional) To use unequaly length bays:
   - a. Click the [...] adjacent to the **No. of bays along length** field.
      A dialog opens with the length of each bay.
b. Type the length of the different bays in the corresponding **Length** cells.

**Note**: The total length of the bays must add to the overall length specified in the **Select Parameters** dialog. The precision should be to about 3 decimal places (e.g., 11.333).

c. Click **OK**.

**7.** Type the **No. of bays along width** to define the number of transverse bays.

**8.** (Optional) To use unequal length transverse bays:
   a. Click the […] adjacent to the **No. of bays along width** field.
      A dialog opens with the length of each bay.
   b. Type the length of the different bays in the corresponding **Length** cells.

**Note**: The total length of the bays must add to the overall width specified in the **Select Parameters** dialog. The precision should be to about 3 decimal places (e.g., 11.333).

c. Click **OK**.

**9.** Click **Apply**.

**M. Frame Models**

The process of generating a Frame structural model and combining it with the existing STAAD.Pro structure is the same as for Trusses (see the above description).

**M. To create a frame or continuous beam model**

To create a continuous beam, bay frame, grid frame, or floor grid model, use the following procedure.

**1.** Select **Frame Models** from the **Model Type** drop-down list.

**2.** Select one of the rectangular frame or beam types by either:
   - double-clicking the icon
   - or
   - click-and-drag the icon to the view window

   - **Bay Frame** – a 3D frame with no horizontal connecting members at the base
   - **Grid Frame** – a 3D frame that has horizontal connecting members at all levels
   - **Floor Grid** – a 2D frame in the XZ plane
Continuous Beam – a linear beam continuous over internal supports (i.e., the bay length)

The Select Parameters dialog opens.

3. (Optional) Type a Model Name.
4. Type the overall dimensions of the frame or beam:
   a. Type the Length.
   b. For bay frames and grid frames, type the Height.
   c. For frames, type the Width.
5. Type the number of bays in each dimension:
   a. Type the No of bays along the length.
   b. For bay frames and grid frames, type the No of bays along the height.
   c. For frames, type the No of bays along the width.
6. (Optional) To use unequal length division in any direction:
   a. Click the [...] button adjacent to that field.
      A dialog opens.
   b. Type the length of each bay.
   c. Click OK.

   Note: The total length of the bays must add to the overall length specified in the Select Parameters dialog. The precision should be to about 3 decimal places (e.g., 11.333).
7. Click Apply.

M. To create a cylindrical frame or beam

To create a cylindrical frame or circular beam, use the following procedure.

1. Select Frame Models from the Model Type drop-down list.
2. Select one of the cylindrical frames or beam types by either:
   double-clicking the icon
   or
   click-and-drag the icon to the view window

   Cylindrical Frame — a cylindrical frame with the cylinder height along the Z axis
   Reverse Cylindrical Frame — a cylindrical frame with the cylinder height along the Y axis
   Circular Beam — a single layer of coordinates which form a beam along a circular arc (i.e., polar coordinates)

The Select Parameters dialog opens.
3. (Optional) Type a **Model Name**.
4. For frames, type the total **Length** along the central axis of the cylinder.
5. Type the **Radius** and the sweep **Angle** of the frame.

   **Tip:** For a full circular frame or beam, leave the Angle 360.

6. For frames, type the **No. of bays along length**.
7. (Optional) For frames with irregular spaced bays along the length:
   a. Click the […] button adjacent to the **No. of bays along length** field. A dialog opens.
   b. Type the length of each bay.
   c. Click OK.

   **Note:** The total length of the bays must add to the overall length specified in the **Select Parameters** dialog. The precision should be to about 3 decimal places (e.g., 11.333).

8. Type the **No of bays along periphery** to specify the number of spaces along the circumference of the circle.
9. (Optional) For frames with irregular spaced bays along the length:
   a. Click the […] button adjacent to the **No of bays along periphery** field. A dialog opens.
   b. Type the angle subtended by each bay.
   c. Click OK.

   **Note:** The total of the angles of all the bays must sum to equal the total Angle given in the **Select Parameters** dialog.

10. Click **Apply**.

**M. Surface or Plate Models**

The surfaces like Quadrilateral Plate, Cylindrical Surface, Hyperbolic Paraboloid Shell, Polygonal plate with Holes and Circular plate with Holes can be created and meshed with the given parametric values.

**M. Quadrilateral Plate**

The Quadrilateral prototype can be used to mesh a quadrilateral surface into small plate elements. Select the **Quadrilateral** prototype under the model type **Surfaces**. Drag the item into the right-side window and release the button. The **Select Meshing Parameters** dialog box will appear to specify the parameters.
Corners

Provide the relative coordinates of the corners of the Quadrilateral Surface or region you want to mesh.

Bias & Divisions

Specify the number of divisions you want along all the edges and the respective biasing. The minimum and maximum limits of number of divisions of each side are 1 and 100 respectively. Two opposite sides may have different numbers of divisions. When the number of divisions for two opposite sides are different, the sum of all divisions must be an even number for quadrilateral elements.

Bias

If you want equal divisions along the length of a side, keep the Bias as 1. If your intention is to divide a side in such a way that the length of the last division is five times the length of the first division, specify Bias for that side as 5 along with the number of divisions.

Biasing may be negative. When negative Biasing has been specified, the side is divided so that the first division length is the value of the Biasing times the last division length.

Element Type

Select the appropriate radio button depending on whether you want to mesh the region in smaller Triangular Elements or Quadrilateral Elements.

M. Cylindrical Surface

Select the Cylindrical Surface prototype under the model type Surfaces. Drag the item into the right-side window and release the button. The Select Parameters dialog box will appear to specify the parameters as shown in the next figure.
The Cylindrical Surface can be generated by providing the basic geometrical parameters **Length**, **Start Radius**, **End Radius** and **Sweep Angle**. You can generate a tapered cylinder by providing unequal values at **Start** and **End radius**. Entering a **Sweep Angle** less than 360 degree can also generate a longitudinally cut cylinder.

The **Axis** of the cylinder can also be controlled by entering the values of **Start** - X, Y, Z values and **End** - X, Y, Z values. The direction of the line joining the points gives the axis of the cylinder.

Providing the number of **Divisions along Length** and **Divisions along periphery**, the mesh can be generated. The option, **Generate open cylinder** will keep the ends open and **Generate triangular element** will generate triangles instead of quadrilaterals.

**M. Hyperbolic Paraboloid Shell**

Select the **Hyperbolic Paraboloid Shell** prototype under the model type **Surfaces**. Drag the item into the right-side window and release the button. The **Select Meshing Parameters** dialog box will appear to specify the parameters.

The basic geometrical parameters for generating a Hyperbolic Paraboloid Surface are the length of the two adjacent sides and the **Corner Dips**. The meshing parameters are the divisions along the adjacent edges. Click **Apply** after specifying all the parameters to generate the model.

![Select Meshing Parameters](image)

**M. Spherical Surface**

A Spherical Surface can be generated and meshed with this feature. Three types of Spherical Surfaces may be generated, viz., **Spherical Cap**, **Spherical Surface** and **Spherical Region**. The **Diameter of Sphere** is required for all those cases. **Base Diameter** is required for Cap and Region only. **Top Diameter** is required for Spherical Region only. For Spherical Surface you have to define the **Start Angle** and **End Angle** in degrees for both the directions - **Latitude** and **Longitude**.

For the purpose of mesh generation, the number of **Divisions** in both the directions is to be specified. Click **Apply** after specifying all these parameters to generate the model.

The **Select Meshing Parameters** dialog box is shown below in the case of a Spherical Surface.
**M. Polygonal Plate with Holes**

Select the **Polygonal Plate with Hole** prototype under the model type **Surfaces/Plates**. Drag the item into the right-side window and release the button. The **Define Meshing Region** dialog box will appear as shown below.

The Polygonal prototype allows us to mesh a polygonal surface, with or without different kinds of holes inside the boundary, into small triangular plate elements.

**Boundary**

The Boundary tab as shown above in the **Define Mesh Region** dialog box provides us an option to specify the **location of Corners, Number of Divisions** of each side and Bias of each side's division for the Boundary to mesh the surface.

X, Y, Z are the coordinates of the various corners of the surfaces. The sides will be defined as joining the various corners. **Div.** option is to define the divisions of various sides of the polygonal surface to be considered for generating the surface meshing. The **Bias** option helps the user to create divisions having unequal spacings among them.
On the top of the right side of this dialog box there are two icons namely, **Add New Row** and **Delete Row**. Clicking on the first icon will add a new row enabling us to specify the coordinates of the corner of the polygonal surface. Clicking on the second icon will enable us to delete an existing row.

**Holes**

After creating the boundary, the user can start introducing the holes in the plate. In STAAD, we may create circular, elliptical as well as polygonal holes.

When we click on the **HOLEs** option, two icons appear as shown below namely, **Add New Hole** and **Delete All Holes**.

- **Add New Hole**
  - This icon allows us to add new holes to the surface.

- **Delete All Holes**
  - This icon allows us to remove all the existing holes from the surface.

  Once a new hole is added, the name of the that hole appears under **HOLEs** in a tree view. A new icon titled **Rename Hole** will now appear which will enable us to rename the newly created hole.

**Hole**

To create a hole within the boundary, click this option. This is the default name of the hole. When we click on Hole, a table will be displayed on the right side of the dialog box, where we can define the geometry of the hole. Specify what kind of hole we want to create from among the three available options namely Polygon, Circle and Ellipse, in the **Region Type**, as shown in the next figure. For each hole, one tab will be created automatically in the **Define Meshing Region** dialog box to allow us to specify different parameters for that hole.

The creation and editing of the region types are explained below.

**Polygon**

This option is used to define polygonal holes. The geometry of the polygonal holes is generated in the same way as the geometry of the polygonal. Please refer to the description under polygonal surfaces for details on defining the geometry of polygonal holes.
Circle  This option is used to define circular holes. The parameters for defining the geometry of circular holes are described below:

Origin  We have to specify the X, Y, and Z coordinates in its proper places to define the position of the center of the circular hole.

Radius of circle  The value in this box will define the radius of the circular hole.

Division Along Periphery  The value in this box defines the number of divisions to be done along the circumference of the circular hole. This information will be considered in mesh generation of the surface.

Ellipse  This option is used to generate ellipsoidal holes. The parameters to define the geometry of ellipsoidal holes are as follows:
Origin
We have to specify the X, Y, and Z coordinates in its proper places to define the position of the center of the ellipsoidal holes.

Major Axis Radius, Minor Axis Radius
The values in these boxes will define the major axis and the minor axis of the ellipsoidal holes.

Division of each Quadrant
The values specified in this box specifies the number of divisions of each quadrant. This information will be used for the mesh generation of the surface.

Once all the parameters have been specified, clicking on OK to carry out the mesh generation operation of the surface. Clicking on Cancel will cancel the mesh generation operation. Clicking on Reset will reset the parameters provided at the beginning.

Note: The one limitation of the Polygonal Surface Meshing is that the total number of nodes must not exceed 15000.

M. Circular Plate with Hole
Circular Plate with hole is a special case of Polygonal Plate with Hole, where the Boundary is circular, keeping the other parameters unaltered.

Select the Circular Plate with Holes prototype under the model type Surfaces/Plates. Drag the item into the right-side window and release the button. The Define Meshing Region dialog box will appear as shown in the next figure.

The basic parameters required to define the circular plate geometrically are the location of the center (Origin) and Radius. To specify the plane of the circular plate the Normal Vector can also be defined. As the meshing parameter, Number of Division along Periphery is required.
M. Cooling Tower

A hyperbolic paraboloid shaped Cooling Tower may be generated and meshed with the parameters displayed in the Select Meshing Parameters dialog box as shown in the next figure.

The basic geometrical parameters required to model such a structure are Top Diameter, Throat Diameter, Height of Tower and Distance of Top from Throat. The Base Diameter is internally calculated and displayed to the user. For the purpose of mesh generation, the number of Division along Circumference and Division along Height is to be specified.

A complete model of a Cooling Tower is shown below.
M. Solid

A Solid rectangular block can be generated and meshed with the parameters displayed in the Select Meshing Parameters dialog box as shown below.

The required parameters are the Lengths along X, Y and Z directions and the corresponding Number of Divisions along those edges.

A Solid rectangular block is shown below.
M. Composite Model: Bunker or Silo

A Bunker or Silo can be modeled parametrically and meshed with this feature.

Select the Bunker or Silo prototype under the model type Composite Models. Drag the item into the right-side window and release the button. The Select Meshing Parameters dialog box will appear as shown below.
To define the geometry of the Vertical Portion, you have to define the **Length**, **Width** and **Height** of that portion along-with the number of **Divisions** along them for the purpose of mesh generation.

The **Depth of the Hopper Bottom** is required with the number of Divisions along depth. The **Length** and **Width** of the opening at the bottom is required and to locate this opening the **shift of Centroid** of the opening w.r.t. the axis of symmetry of the Silo is required.

Checking the box **Enable Edge Beam Generation** can generate edge beams and also checking the box **Enable Corner Column Generation** can generate Corner Columns, for which the **Height of Column from Top** is to be provided.

### M. Import CAD Model

This feature, Import CAD Model, has two separate utilities, Scan DXF and STAAD Models.

#### M. To generate geometry from a DXF file

This is a utility which allows the geometry (nodes, members, plates, etc.) of a previously created model to be imported and altered.

This feature supports a limited number of CAD entities like Line, 3D-Polyline, and 3D-Face.

If the geometry of a model is created using a drawing program like AutoCAD and saved in a DXF file, it can be imported into STAAD using this utility.

1. **Select Import CAD Models** from the **Model Type** drop-down list.
2. **Either:**

   Drag-and-drop the **Scan DXF** icon into the view window.

   or
Double-click the **Scan DXF** icon.
The **Open** dialog opens.

3. Select the DXF file you want to use and click **Open**.
The drawing geometry is opened in Structure Wizard.

### M. To generate geometry from a STAAD model

This feature can be used to import an existing model which can then be quickly scaled up or down in the Structure Wizard.

1. Select **Import CAD Models** from the **Model Type** drop-down list.
2. Either:
   - Drag-and-drop the **STAAD Model** icon into the view window.
   - or
   - Double-click the **STAAD Model** icon.
The **Open** dialog opens.
3. Select the STAAD input file (`.std`) you want to use and click **Open**.
The model geometry is opened in Structure Wizard.

### M. VBA Macro Models

This feature allows you to parametrically define models using a VBA Macro.

Two sample files are supplied with the program.

**Tip:** Select **File > Set as Top Most** to turn off that feature before launching a macro. Otherwise, the macro form may be hidden beneath the **Structure Wizard** application window.

- Dragging a macro icon from the left side of the window to the right to launch the macro.
- Double-click an icon on the left-hand side to open the code in the VBA editor. You can then make changes and save the file to suit your needs.

**Related Links**
- *EX. Macro Example Files* (on page 4310)

### M. To add a new plugin

1. Either:
   - Select **File > Add Plugin**
   - or
   - Right-click in the model types list and select **Add Plugin** from the pop-up menu.
The **Select an existing Macro** dialog opens.
2. Select the Visual Basic Macro file (file extension `.vbs`) you want to add and click **Open**.
The new macro is added to the list of plugins in the model types list.
M. To edit model parameters

1. Either:
   - Right-click in the view window and select **Change Property** from the pop-up menu.
   - or
   - Select **Edit > Change Property**.
   - or
   - Double-click on the model view window.
   - The **Select Parameters** dialog opens.

2. Change one or more parameters as needed.

3. Click **Apply**.

M. To rescale a model

1. Either:
   - Right-click in the view window and select **Scale** from the pop-up menu.
   - or
   - Select **Edit > Scale**.
   - The **Rescale Model** dialog opens.

2. Type in corresponding Z, Y, and Z scale factors in the fields.

3. Click **OK**.

M. To delete an entire model

Right-click in the view window and select **Delete** from the pop-up menu

M. To transfer the generated model to STAAD.Pro

1. Select **File > Merge Model with STAAD.Pro Model**.
   - A dialog prompts you to confirm you want to merge this prototype with your current STAAD.Pro model.

2. Click **Yes**.
   - The **Paste Prototype Model** dialog opens.
3. (Optional) Specify the method of offset:

**Tip:** This is useful, for example, to place a prototype truss model at the top of already modeled columns.

<table>
<thead>
<tr>
<th>Move option</th>
<th>Do the following</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>By distance between following two nodes</strong></td>
<td>Type the node numbers</td>
</tr>
<tr>
<td><strong>By the following X, Y and Z values</strong></td>
<td>Type the coordinates (in the indicated units) where to place the prototype origin in the global coordinates of the STAAD model.</td>
</tr>
</tbody>
</table>

4. Click **OK**. The model is placed.

In the event that nodes, members, or elements overlap existing similar objects in the STAAD model, a message dialog opens indicating that the duplicate objects have been ignored. Click **OK** to proceed.

**M. To add items to the library**

You can customize the list of model types using the following procedure.

1. Select the **Model Type** from the drop-down list.
2. Either:
   - Select **File > Add Plugin**
   - or
   - right-click in the model type list and select **Add Plugin** from the pop-up menu
   - The **Open** dialog opens.
3. Select the library file (file extension .dll) for the plugin and click **Open**.
4. (Optional) To rename a selected model generator, do the following:
   a. Select a model generator and either:
      - Select **Edit > Rename Model Generator**
or

right-click on the model generator and select **Rename Model Generator** from the pop-up menu

The **Select Name** dialog opens.

b. Type a new name.

c. Click **OK**.

5. (Optional) Save the library of prototypes to a new file:

a. Select **File > Save As**.

The **Save As** dialog opens.

b. Type a new file name for the StWizard Model Files (file extension .stp) and click **Save**.

c. You can now open this file again by selecting **File > Open**.

---

**M. Pages in the Analytical Modeling Workflow**

- **Geometry** – used to layout your structure, including nodes, beams, plates, and solids

  **Note:** If you are using the Physical Modeling workflow, the majority of the tools on this page will be disabled. Model geometry for physical models must be updated in the STAAD.Pro Physical Modeler interface.

- Properties
- Specifications
- Supports
- Loading
- Analysis
- Design

---

**Nodes table**

Used to display a table of nodes with coordinates for adding and editing nodes. The table may also be used to highlight the node on the graphical view of the structure.

The **Nodes** table lists the node numbers and their coordinates. This table can be used to define new nodes and edit existing node-coordinates. If we click on a node number in this table, the node gets highlighted in the table as well as in the structure view.

Opens when the **Geometry** page is selected in the **Analytical Modeling** workflow.

- **Node** Lists the node numbers in numerical order.

  - **X** The X coordinates of the nodes in the currently selected unit. Changing this immediately changes the X-coordinate of the node in the graphical display also.

  - **Y** The Y coordinates of the nodes in the currently selected unit. Changing this immediately changes the Y-coordinate of the node in the graphical display also.

  - **Z** The Z coordinates of the nodes in the currently selected unit. Changing this immediately changes the Z-coordinate of the node in the graphical display also.

---

**Goto <object> dialog**

Used to go to a specific node within the **Nodes, Beams**, etc. table.

Opens when **Go To** is selected from the right-click pop-up menu on many tables.
Type the number of a node, beam, etc. and then click **OK**. The table will highlight that entry.

**Node Supports**
Lists all nodes for which supports have been defined along with the type of support.

---

### M. Properties and Specifications

#### M. Section Profiles

Beam members must have a section profile assigned for analysis.

STAAD.Pro supports a wide variety of section profiles, including catalog sections, prismatic sections, tapered member sections, and so on.

**M. To add a new table section property**

To add a table section property from a standard catalogs for steel, cold-formed steel, timber, or aluminum, use the following procedure.

These are also referred to as the “built-in” tables of sections in STAAD.Pro.

1. **Either:**
   - select the Properties page and then click **Section Database** on the Properties - Whole Structure dialog
   - or
   - select the Database tool in the Specifications group on the Specifications ribbon tab

   The **Section Profile Tables** dialog opens.

2. Select the appropriate table:
   a. Select the material tab at the top of the dialog:
      - **Steel**
      - **Coldformed Steel**
      - **Timber**
      - **Aluminium**
   b. Select the country catalog and shape profile on the left-hand side of the dialog.

   **Tip:** If you selected the material and country using the Database tool drop-down, then only that material and country tables are shown in the dialog.

3. Select the shape profile on the left-hand side of the dialog.
4. **Either:**
select the specific shape (i.e., beam, profile, shape, section, etc.) in the list

or

select to define a shape for some (i.e., pipes)

**Note:** Refer to related codes in the Design section (on page 944) for information on section nomenclature.

5. Select the appropriate **Type Specification** for some shapes.

Some type specifications require additional input (such as cover plate dimensions, spacing, composite slab data, etc.).

6. **(Optional)** Select an associated **Material**.

This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.

7. Either:

   **To...**

   add the section to the model for later assignment  

   add the section to the model and assign to the current member selection

   **Do the following...**

   click Add.

   click Assign.

8. **(Optional)** Repeat steps 2 through 7 as needed to add more sections to the model.

9. Click **Close**.

The section and, if selected, associated material, is added to the **Properties - Whole Structure** dialog.

If you did not assign the section to a selection set, you must select the section label in the **Properties - Whole Structure** dialog and assign it using one of the assignment methods.

**Related Links**

- **G.6.2 Built-In Steel Section Libraries** (on page 2325)
- **Properties - Whole Structure dialog** (on page 2968)
- **TR.20 Member Property Specification** (on page 2459)
- **Section Profile Tables dialog** (on page 2969)
- **TR.20.1 Assigning Properties from Steel Tables** (on page 2461)

**M. To add an American steel joist section**

To add an American steel joist section and, optionally, have it sized by span and load, use the following procedure.

STAAD.Pro includes the facilities for specifying steel joists and joist girders. The basis for this implementation is the information contained in the 1994 publication of the American Steel Joist Institute called "Fortieth edition standard specifications, load tables and weight tables for steel joist and joist girders." Member properties can be assigned by specifying a joist designation contained in tables supplied with the program, including the following joists and joist girder types:

- Open web steel joists – K series and KCS joists
- Longspan steel joists – LH series
- Deep Longspan steel joists – DLH series
- Joist Girders – G series

**Tip:** It is recommend that use the default selection of **Steel** material definition with joists.
1. Either:
   select the Properties page and then click Section Database on the Properties - Whole Structure dialog
or
   select the Database tool in the Specifications group on the Specifications ribbon tab

The Section Profile Tables dialog opens.

2. Select the Steel tab and then the American Steel Joist entry on the left-hand side.

3. Select the series of joists you want to use.

4. (Optional) To have the program select a least-weight option of joist for you, do the following:
   a. Type the search criteria (differs by joist series) in the Find suitable joist section fields.
   b. Click Find Section.
   For example, if you select the LH series of joists and then specify a Clear Span of 35 ft, a Depth Limit of 24 in
      (2ft), a Total Load of 750 lbs/ft (0.75k/ft), a Live Load of 500 lbs/ft (0.5k/ft), and a Deflection Limit of L/360,
      the program will select a 24LH09 when you click Find Section. The least weight suitable joist in the joist series
      will be selected. If not suitable joist can be found in the series, a message dialog opens.

5. Either:
   To... Do the following...
   add the section to the model for later assignment click Add.
   add the section to the model and assign to the current member selection click Asign.

6. Click Close.

Example Input
An example of a structure with joist:

STAAD SPACE EXAMPLE FOR JOIST GIRDER
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0
3 30 10 0; 4 30 0 0
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W21x50
MEMBER PROPERTY SJIJOIST
2 TABLE ST 22K6
CONSTANTS
E STEEL ALL
DENSITY STEEL ALL
POISSON STEEL ALL
SUPPORTS
1 4 FIXED
UNIT POUND FEET
LOAD 1
SELFWEIGHT Y -1
LOAD 2
### Member Load

<table>
<thead>
<tr>
<th>MEMBER LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 UNI GY -250</td>
</tr>
<tr>
<td>LOAD COMB 3</td>
</tr>
<tr>
<td>1 1 2 1</td>
</tr>
<tr>
<td>PERF ANALY PRINT STAT CHECK</td>
</tr>
<tr>
<td>PRINT SUPP REAC</td>
</tr>
<tr>
<td>FINISH</td>
</tr>
</tbody>
</table>

**Related Links**

- *Section Profile Tables dialog* (on page 2969)

### M. To assign a prismatic section

To define and assign a prismatic section, use the following procedure.

1. **Either:**
   - select the **Properties** page and then click **Section Database** on the **Properties - Whole Structure** dialog
   - or
   - select the **Database** tool in the **Specifications** group on the **Specifications** ribbon tab

   The **Section Profile Tables** dialog opens.

2. **Either:**
   - select the Properties page and then click **Define** on the **Properties - Whole Structure** dialog
   - or
   - on the **Specifications** ribbon tab, select the **Prismatic** tool in the **Beam Profiles** group

   The **Property** dialog opens.

3. **Select the tab corresponding to the profile shape you want to use.**

   **Notes:**
   - The geometric shapes circle, rectangle, tee, trapezoidal, and general are typically used for concrete members.
   - Tapered I and tapered tube are typically used for steel members.
   - The Assign Profile tab is used to assign a profile shape for steel design for later member selection.

4. **Type the profile dimensions for the selected shape.**

5. **(Optional) Select an associated **Material**.**
   - This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.

6. **Either:**
To... Do the following...
add the section to the model for later assignment click Add.
add the section to the model and assign to the current member selection click Assign.

7. Click Close.

Related Links
• G.6.1 Prismatic Properties (on page 2323)
• Property dialog (on page 2970)
• TR.20.2 Prismatic Property Specification (on page 2465)
• TR.20.2.1 Prismatic Tapered Tube Property Specification (on page 2467)

M. To assign a tapered I section

To define and assign a web-tapered I section, use the following procedure.

Note: $f_1$ (Depth of section at start node) should always be greater than $f_3$ (Depth of section at end node). You must provide the member incidences accordingly.

Tapered I-sections have constant flange dimensions and a linearly varying web depth along the length of the member.

Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the SET SHEAR command has no effect on the deformation computed for members with these cross sections.


The Tapered I dialog opens.

2. Select an associated Material.

This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.

Note: Tapered I sections are typically steel or aluminum members.

3. Specify the profile dimensions for the shape.
4. Either:
   
   **To...**
   
   add the section to the model for later assignment
   
   add the section to the model and assign to the current member selection
   
   **Do the following...**
   
   click Add.
   
   click Asign.

5. Repeat steps 2 through 4 as needed to add more tapered I sections to the model.

6. Click Close.

The section and, if selected, associated material, is added to the Properties - Whole Structure dialog.

If you did not assign the section to a selection set, you must select the section label in the Properties - Whole Structure dialog and assign it using one of the assignment methods.

**Related Links**

- [G.6.4 Tapered Sections](on page 2326)
- [TR.20.3 Tapered Member Specification](on page 2468)
- [Property: Tapered I dialog](on page 2972)

**M. To create a general section**

To create a user provided table section with a general profile, use the following steps.

You may want to change the input dimensions prior to creating user provided table sections. Select **Tools > Set Current Input Unit** to do so.

1. On the Specification ribbon tab, select the User Table > User Table Manager tool in the Beam Profiles group.

   ![User Table Manager]

   If no User Defined Table exists, you will be prompted to create one.

   The User Table Manager dialog opens.
2. Click New Table.
   The New User Table dialog opens.

3. In the Select Section Type list, select GENERAL and then click OK.
   The dialog closes and the Select Existing Table list in the User Table Manager dialog now has at least one entry.

4. Click Add.
   The General dialog opens.
5. Type a **Section Name**.

6. Either:

   Type the section properties if these are available in the corresponding fields
   or

   Set the **Define Profile Polygon** and enter in Z,Y coordinates of the section apexes.

   **Tip:** These values may be copied and pasted from a spreadsheet.

   The following is an example cross section, with corresponding Z,Y coordinates which can be copy/pasted directly into the table.
### Coordinate Number Table

<table>
<thead>
<tr>
<th>Coordinate Number</th>
<th>Z (units)</th>
<th>Y (units)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>-4</td>
</tr>
<tr>
<td>3</td>
<td>3.5</td>
<td>-4</td>
</tr>
<tr>
<td>4</td>
<td>3.5</td>
<td>-16</td>
</tr>
<tr>
<td>5</td>
<td>6.5</td>
<td>-16</td>
</tr>
<tr>
<td>6</td>
<td>6.5</td>
<td>-4</td>
</tr>
<tr>
<td>7</td>
<td>13.5</td>
<td>-4</td>
</tr>
<tr>
<td>8</td>
<td>13.5</td>
<td>-16</td>
</tr>
<tr>
<td>9</td>
<td>16.5</td>
<td>-16</td>
</tr>
<tr>
<td>10</td>
<td>16.5</td>
<td>-4</td>
</tr>
<tr>
<td>11</td>
<td>20</td>
<td>-4</td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>0</td>
</tr>
</tbody>
</table>

7. (Optional) If you entered in section corner coordinates, click **Compute Section Properties**

The section properties are calculated and the fields are updated. The coordinates are also updated from an arbitrary origin to the an origin at the center of gravity of the cross section.
Tip: You can manually overwrite any of these values if necessary.

8. (Optional) If you want to have section stress calculated, enter Y,Z coordinates in the **Stress locations in local coordinates**

Tip: The coordinate locations for stresses should be based on the center of gravity of the section, rather than an arbitrary data point. These may be copied/pasted from the re-centered apex coordinates table.

9. Click **OK**.

The section is added to the list in the **Create User Property Table** dialog.

10. (Optional) Repeat steps 4 through 9 to add more General sections.

11. Click **Close**.

The section can now be added for use in the **Properties - Whole Structure** dialog (on page 2968) by selecting it in the **User Property Table** dialog.

**Example**

The example given in 6 above creates the following lines in a STAAD input file:

```plaintext
... TABLE 1 UNIT INCHES KIP GENERAL General1 152 16 0 20 0 3395.93 4520.67 638.828 332.591 452.067 83.3308 - 46.3944 760 625.529 95637.5 16 PROFILE_POINTS -10 5.78947 -10 1.78947 -6.5 1.78947 -6.5 -10.2105 -3.5 -10.2105 -3.5 1.78947 3.5 1.78947 -3.5 -10.2105 6.5 -10.2105 6.5 1.78947 10 1.78947 10 5.78947 STRESS_LOCATIONS -10 5.78947 10 5.78947 -6.5 -10.2105 6.5 -10.2105 END ...```

**Related Links**

- G.6.3 User-Provided Steel Table (on page 2325)
- TR.19 User Steel Table Specification (on page 2446)
- TR.20.4 Property Specification from User Provided Table (on page 2469)
- Create User Provided Table dialog (on page 2974)
- User Provided Table dialog (on page 2973)

**To use a general shape created in Section Wizard**

To use a general shape created in Section Wizard as a user provided shape table in STAAD.Pro, do the following.

You must create a profile outline in Section Wizard and export it as a General Shape (file extension .upt) for use in STAAD.Pro. Refer to TR.20.10 Member Property Reduction Factors (on page 2485) for details.

**Note:** This profile shape should consist of only an outer contour. Any internal voids such as internal contours or openings will be ignored, though section properties calculated by Section Wizard based on sections with voids will be used.
1. On the **Specification** ribbon tab, select the **User Table > User Table Manager** tool in the **Beam Profiles** group.

   ![User Table Manager](image1)

   If no User Defined Table exists, you will be prompted to create one.

   The **User Table Manager** dialog opens.

2. Click **New Table**.

   The **New User Table** dialog opens.

   ![New User Table](image2)

3. Specify the new external table:
   a. In the **Select Section Type** list, select **GENERAL**.
   b. Check the **External Table** option.
   c. Click **Browse**.
   d. Navigate to and select the '.upt' file exported from Section Wizard and the click **OK**.

   The dialog closes and the **Table Data** in the **User Table Manager** dialog shows the name of the profile read from the '.upt' file.
4. (Optional) To view the calculated section properties, select the profile entry in the **User Table Manager** dialog and then click **View**. The **General** dialog opens.

5. Click **Close**.

The section can now be added for use in the **Properties - Whole Structure** dialog (on page 2968) by selecting it in the **User Property Table** dialog.

**Tip:** Changing the structure diagram to display 3D rendering types for Full Sections or Sections Outline will display the shape profile. Similarly, 3D renderings of the structure will also display the shape profile.
M. To assign catalog section to physical members

To select and assign a catalog section to physical members, use the following procedures.

**Note:** For a physical member whose physical member property is already assigned, the individual analytical members in that Physical member will adopt the same member property that of the physical member. However, in case, where the analytical member property is assigned to any member in that physical member, then this analytical member property will supersede physical member property.

1. Select the Properties page.
   The **Properties - Whole Structure** dialog opens.
2. On the **Geometry** ribbon tab, select the **Toggle Physical Member Mode** tool in the **Physical Members** group.
   
   The tool is highlighted to indicate the mode is active.
3. Either:
   select the **Properties** page and then click **Section Database** on the **Properties - Whole Structure** dialog
or

select the Database tool in the Specifications group on the Specifications ribbon tab

The Section Profile Tables dialog opens.

4. Select the appropriate table:
   a. Select the material tab at the top of the dialog:
      - Steel
      - Coldformed Steel
      - Timber
      - Aluminium
   b. Select the country catalog and shape profile on the left-hand side of the dialog.

   **Tip:** If you selected the material and country using the Database tool drop-down, then only that material and country tables are shown in the dialog.

5. Select the shape profile on the left-hand side of the dialog.

6. Either:
   - select the specific shape (i.e., beam, profile, shape, section, etc.) in the list
   - or
   - select to define a shape for some (i.e., pipes)

   **Note:** Refer to related codes in the Design section (on page 944) for information on section nomenclature.

7. Select the appropriate Type Specification for some shapes.
   - Some type specifications require additional input (such as cover plate dimensions, spacing, composite slab data, etc.).

8. (Optional) Select an associated Material.
   - This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.

9. Click Add.
   - The new member property is added to the list as a Physical member property in the Properties - Whole Structure dialog. Such properties are designated with the word “Physical”.

   **Note:** Only the Physical Member properties can be assigned to a physical member, and no analytical member property can be assigned to a physical member.

10. Click Close.
    - You must assign the physical member properties to physical members using one of the standard assignment methods from the Properties - Whole Structure dialog.

   **Related Links**
   - *M. Physical Members* (on page 649)

   **M. Section Wizard Help**
M. To start Section Wizard

Section Wizard consists a set of utility programs used to generate custom sections for use in STAAD.Pro.

1. Open a model in the STAAD.Pro Analytical Modeling workflow.
2. On the Specifications ribbon tab, select Section Wizard tool in the Tools group.

The Section Wizard program opens.

Alternatively, you can locate the file \SectionWizard\Section.exe in the folder where STAAD.Pro was installed on your computer.

M. Section Builder

The Section Builder module is used to calculate the properties of sections built up from an arrangement of cross sections taken from a number of standard tables. This module is composed of two windows. The main Section Builder window, which displays the composite section along with a number of tools to manipulate the sections and the Section Element dialog box which is used to select the sections to be used and method of locating them onto the current overall section.

M. Application Window Layout

The main Section Builder window displays the current composite section. All of the primary commands are activated through the menus along the top of the window. There is also a status bar along the bottom of the window which displays the overall dimensions of the composite section and the location of the cursor when it lies over the main workspace. Additionally a dimension value will be displayed in the status bar by clicking and dragging the mouse across the window.

Sections created in the Section Dialog box can be located onto a composite section relative to the nodes of the 'current section.' A section is made into the current section simply by clicking on it so that it is shown in yellow. Note how the nodes of the current section are identified as potential locating points for the addition of new sections.
Window controls

Menu

Toolbar  A series of commonly used tools are arranged at the top of the module window.

M. Toolbar
Contains several of the most commonly used tools found in the Section Builder module.

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>----------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Open</td>
<td>Opens the Open Data File dialog, which is used to select either Section Builder (file extension .SEC) files or Free Sketch (file extension .CNS) files. Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the Preview option is selected, then a graphical preview of the section in a selected file is displayed prior to opening the file.</td>
<td></td>
</tr>
<tr>
<td>Save</td>
<td>Opens the Save dialog box if the file has not previously been saved and allows the Equivalent Section, section to be saved. If the file has previously been saved, then that file is updated with the new data. Files are saved in the Section Builder *.SEC format.</td>
<td></td>
</tr>
<tr>
<td>Create Standard Section</td>
<td>Opens the Section dialog (on page 762) which is used to quickly generate a composite section based on one of seven options. This is the same option that is available in the Section Builder module.</td>
<td></td>
</tr>
<tr>
<td>Undo</td>
<td>Negates the last drawing operation.</td>
<td></td>
</tr>
<tr>
<td>Redo</td>
<td>Negates the last undo operation.</td>
<td></td>
</tr>
<tr>
<td>Delete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shift/Rotate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Copy</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shift Coordinate Center</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Calculate Basic Geometry</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>------------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Report</td>
<td>Once an equivalent section has been 'found', the data displayed in the table and a dimensioned drawing will be sent to a report formatted to the Report Type option defined in the Misc sheet of the Program Preferences dialog. See Preferences below.</td>
<td></td>
</tr>
<tr>
<td>Preview</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Show Coordinate Axis</td>
<td>Toggles the display of the coordinate axis in the Section Builder view.</td>
<td></td>
</tr>
<tr>
<td>Show Grid</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Show Principle Axis</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Show Current Center of Gravity</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Zoom In</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Zoom Out</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Preferences</td>
<td>Opens the Preferences dialog.</td>
<td></td>
</tr>
<tr>
<td>Display Normal Stress Field</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Launch Free Sketch module</td>
<td>Launches the [c_Free_Sketch_module](on page 763) of Section Wizard and creates section based on the section that was 'found'.</td>
<td></td>
</tr>
<tr>
<td>Launch Equivalent Section module</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**M. Menus**

Menus in the Section Builder module window.

**M. File menu**

Contains items for creating, opening, and closing section data files, printing, and exiting the program.
<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Opens an empty section design file. If the currently loaded section has been modified, then the option is given to save it prior to comencing with the new design.</td>
<td>CTRL+N</td>
</tr>
<tr>
<td>Open</td>
<td>Opens the Open Data file dialog, which is used to select an existing Section Data file (file extension .SEC). If the Show Drawing option is checked, then a preview of the section can be displayed prior to opening the file.</td>
<td>CTRL+O</td>
</tr>
<tr>
<td>Create Standard Section</td>
<td>Opens the Section dialog (on page 762) which is used to quickly generate a composite section based on one of seven options.</td>
<td></td>
</tr>
<tr>
<td>Save</td>
<td>Saves any changes made to the current data file.</td>
<td>CTRL+S</td>
</tr>
<tr>
<td>Note: If the current file has not been previously saved, this has the same effect as selecting <strong>Save As...</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Save As...</td>
<td>Opens the Save data file dialog which is used to specify a file name and location for the section data file.</td>
<td></td>
</tr>
<tr>
<td>Report</td>
<td>Formats and sends the data to the Report Type option defined in the Misc sheet of the Program Options dialog. See Options below.</td>
<td></td>
</tr>
<tr>
<td>Preview</td>
<td>Displays the section as will be drawn in the report. Toggles the display of the Section Element dialog.</td>
<td></td>
</tr>
<tr>
<td>Send...</td>
<td>Opens an e-mail (using your default e-mail client) and attaches the current data file.</td>
<td></td>
</tr>
</tbody>
</table>
### Menu item | Description | Same effect as selecting...
--- | --- | ---
**Export to STAAD.Pro >** | Select either to export the section as a General or Prismatic section. A Save...<br> If an existing User Provide Table file is selected, the current section will be appended to the end of this table. Refer to the STAAD.Pro Technical Reference for additional information on using section types. |  
**Sections > <list>** | A list of the most recent four section data files. Select one to open that data file. |  
**Exit** | Closes the program. If any changes were made since the last time the data file was saved, a message dialog opens confirming you wish to save those changes. ALT+F4 |  

### M. Edit menu

| Menu item | Description | Same effect as selecting... |
--- | --- | ---|
**Undo** | Negates the last modification performed in the window (for example, shift/rotate a section). You can undo a series of modifications by repeatedly selecting Undo. |  
**Redo** | Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting Redo. |  
**Delete** | Removes the selected section element from the composite section. Delete key |
### Menu item

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shift, Rotate Element</td>
<td>Opens the Element Shift/Rotation dialog which is used to displace the currently selected section by a relative dimension or rotation about its basic node. See section Orientation of Elements for the basic node of each section type used by Section Builder.</td>
<td></td>
</tr>
<tr>
<td>Shift Coordinate Center</td>
<td>Opens the <a href="#">Shift coordinate center dialog</a> (on page 750) which is used to relocate the origin of the geometric axes relative to the composite section.</td>
<td></td>
</tr>
<tr>
<td>Copy Element</td>
<td>Opens the Copy element dialog which is used to create one or more copies by specified offset dimensions in the Y and Z axes.</td>
<td></td>
</tr>
</tbody>
</table>

### M. Shift coordinate center dialog

Used to relocate the origin of the geometric axes relative to the composite section.

**Dialog controls**

1. Set it to the current center of gravity
2. Set it to a given node on the current section.
3. Set it to a given node on the current section.

### M. Settings menu

**Table 33: Settings menu items**

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Redo</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Delete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shift, Rotate Element</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shift Coordinate Center</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Copy Element</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### M. Tools menu
### Table 34: Tools menu items

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculate</td>
<td>Calculates the cross-sectional properties of the composite section and opens the Basic Geometry dialog to display the values.</td>
<td></td>
</tr>
<tr>
<td>Stress Contour</td>
<td>Opens the Section forces dialog, which is used to specify major axis moments and axial force on the section. Click OK to close the dialog and display the stress distribution on the composite section.</td>
<td></td>
</tr>
<tr>
<td>Free Sketch</td>
<td>Opens the [Free Sketch module](page 763).</td>
<td></td>
</tr>
<tr>
<td>Equivalent Section</td>
<td>Opens the [Equivalent Section module](page 757).</td>
<td></td>
</tr>
<tr>
<td>Windows Calculator</td>
<td>Opens the Windows system Calculator window.</td>
<td></td>
</tr>
<tr>
<td>Formula Calculator</td>
<td>Opens the [Formula Calculation window](on page 784) which is used to evaluate a formula, from the numerical result can then be copied back into the Section Builder program.</td>
<td></td>
</tr>
<tr>
<td>Unit Converter</td>
<td>Opens the Unit Converter Window which is used to as a value converter between different preset units of measurement. Select the tab pertaining to the type of measurement and then enter the known value in the appropriate field. Upon pressing &lt;Enter&gt;, all the other fields are updated to equivalent values.</td>
<td></td>
</tr>
</tbody>
</table>

### M. Help menu

The Help menu contains items for using online help.
**Table 35: Help menu items**

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Help Topics</td>
<td>Opens the Section Builder help window.</td>
<td>F1</td>
</tr>
<tr>
<td>About</td>
<td>Opens the About Section Builder window, which displays version and copyright information.</td>
<td></td>
</tr>
</tbody>
</table>

**M. Orientation of Elements**

**Table 36: Standard element shapes and their default orientation and base nodes**

<table>
<thead>
<tr>
<th>Profile Type</th>
<th>Section</th>
<th>Base Node #</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angles</td>
<td>Equal Leg Angle</td>
<td>1</td>
<td><img src="image" alt="Diagram" /></td>
</tr>
<tr>
<td>Profile Type</td>
<td>Section</td>
<td>Base Node #</td>
<td>Nodes</td>
</tr>
<tr>
<td>-------------------</td>
<td>------------------------------------------</td>
<td>-------------</td>
<td>-------</td>
</tr>
<tr>
<td>Unequal Leg Angle</td>
<td></td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>I Sections</td>
<td>I Sections with Tapered Flanges</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td>Profile Type</td>
<td>Section</td>
<td>Base Node #</td>
<td>Nodes</td>
</tr>
<tr>
<td>-------------------</td>
<td>----------------------------------</td>
<td>-------------</td>
<td>---------------</td>
</tr>
<tr>
<td></td>
<td>I Sections with Parallel Flanges</td>
<td>10</td>
<td>10 11 12</td>
</tr>
<tr>
<td>Channels</td>
<td>Parallel Flanged Channels</td>
<td>1</td>
<td>1 2 3</td>
</tr>
</tbody>
</table>

*STAAD.Pro User Manual*
### Modeling

**M. Properties and Specifications**

<table>
<thead>
<tr>
<th>Profile Type</th>
<th>Section</th>
<th>Base Node #</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tapered Flange Channels</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cold Rolled Channels</td>
<td>1</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Diagram of Tapered Flange Channels](image1.png)

![Diagram of Cold Rolled Channels](image2.png)
<table>
<thead>
<tr>
<th>Profile Type</th>
<th>Section</th>
<th>Base Node #</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHS</td>
<td>Circular Hollow Sections</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>Plate</td>
<td>Plate Sections</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>Tee Section</td>
<td>T Sections with Parallel Flanges</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### M. Equivalent Section module

This module is used to find a best fit solution from the table of values of area, inertia, and elastic moduli. The data can either be entered directly into the table, or generated from the Section Builder or Free Sketch modules, or from the Standard Section toolbar command.

#### Section Types

The Equivalent Section module then finds the best fit on one of the following section types:

1. I Section, with independent dimensions for the top and bottom flanges
2. Rectangular Hollow Section, with separate dimensions for the side walls and the top and bottom flanges
3. I Section, with the same sizes for the top and bottom flanges
4. Rectangular Hollow Section, with the same thickness for the flanges and the side walls
5. Channel Section

Each of the values can be weighted for importance by setting a bias for that value. The larger the bias, the more the program will attempt to match that value rather than any of the others.

M. Application Window Layout

![Equivalent Section module window](image)

**Figure 110: Equivalent Section module window**

Window controls

**Toolbar**
A series of commonly used tools are arranged at the top of the module window.

**Properties table**
A list of the reference property values and bias values are entered here to provide data for the cross-section search. Once the Find button is clicked, a set of associate Replacement values and the percent Error will also be displayed, along with a graphical representation of the found equivalent section dimensions.

**Equivalent cross-section**
A group of controls are arranged here depicting general cross-section types. The currently selected type is shown as depressed.

**Find**
Click to perform a search for an equivalent cross-section based on the specified section Values, weighted per the specified Bias values.

**Round dimensions**
Select this option to have the search use rounded dimensional values. If a search has already been performed, then the current dimension results are rounded.

**Exit**
Closes the module. Any unsaved data will be lost.

**Help**
Opens the Section Wizard help window to display the relevant help topic.
**Views panels**  A graphical representation of a section as well as dimensions of the recommended section from a Find action are displayed in the right-hand side of the window.

**M. Toolbar**
A list of tools found in Equivalent Section.

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open</td>
<td>Opens the Open Data File dialog, which is used to select either Section Builder (file extension .SEC) files or Free Sketch (file extension .CNS) files. Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the <strong>Preview</strong> option is selected, then a graphical preview of the section in a selected file is displayed prior to opening the file.</td>
<td></td>
</tr>
<tr>
<td>Save</td>
<td>Opens the Save dialog box if the file has not previously been saved and allows the Equivalent Section, section to be saved. If the file has previously been saved, then that file is updated with the new data. Files are saved in the Section Builder *.SEC format.</td>
<td></td>
</tr>
<tr>
<td>Report</td>
<td>Once an equivalent section has been 'found', the data displayed in the table and a dimensioned drawing will be sent to a report formatted to the Report Type option defined in the Misc sheet of the Program Preferences dialog. See Preferences below.</td>
<td></td>
</tr>
<tr>
<td>Create Standard Section</td>
<td>Opens the <strong>Section dialog</strong> (on page 762) which is used to quickly generate a composite section based on one of seven options. This is the same option that is available in the Section Builder module.</td>
<td></td>
</tr>
<tr>
<td>Preferences</td>
<td>Opens the Preferences dialog.</td>
<td></td>
</tr>
</tbody>
</table>
### Tool Description

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Launch Section Builder module</td>
<td>Launches the Section Builder module of Section Wizard and creates section based on the section that was 'found'</td>
<td></td>
</tr>
<tr>
<td>Launch Free Sketch module</td>
<td>Launches the <a href="#">c_Free_Sketch_module</a> of Section Wizard and creates section based on the section that was 'found'.</td>
<td></td>
</tr>
<tr>
<td>Export STAAD User Database</td>
<td>This allows the current section data to be used to create STAAD External User tables. There are two types of database that can be created, either a Prismatic Section or a General Section. Selecting either command will ask for a name of file to be used. If the filename chosen exists already, then the current section will be added to the bottom of the file.</td>
<td></td>
</tr>
</tbody>
</table>

**M. Preferences dialog**

Used to set preferences for the Equivalent Section module.

**Units Measures tab**
Select the units for angles, section dimensions, section properties, axial forces and moments, as well as the number of decimal places and whether or not to use an exponential form.

Misc. tab

Select the language, Report settings such as the template document, whether the documents should use Word 7 or Word 97 file formats, the paper size and font. Set whether documents should be sent direct to the printer or
into Word for editing. Additional options determine the number of vertices to created by a circle, option to snap to grid and whether the ends of vertices are indicated by a circle or not.

Profile Databases tab

Select the range of databases that are referred to by the program. See Appendix 2: Section Databases for a current list of available database sections.

Other Controls

These controls are displayed at the bottom of the dialog for all tabs.

- **Apply**: Saves any changes made in the dialog.
- **Cancel**: Closes the dialog without saving any changes.
- **Help**: Opens the Section Wizard help window to the relevant topic.

M. Section dialog
Dialog controls

**Section type selectors**

**Cover Plate dimensions** (For I-sections with cover plates only) These fields become active when the I-section with cover plate type is selected. Specify values for \( b_1 \) and \( h_1 \) (width and thickness of top cover plate, respectively) as well as \( b_2 \) and \( h_2 \) (width and thickness of bottom cover plate, respectively).

- **Database**
- **Family**
- **Section**
- **Cancel**
- **OK**

**M. Free Sketch**

Used to create sections with an outer boundary of any shape with a number of internal openings if required.

Free Sketch data files can be saved in either one of two formats. `<filename>.CNS` is a binary file format which is the most compact and preferred way to store files. Alternatively, files can be stored as `<filename>.CON`, stores the file in an ASCII format which can be opened and edited by any text editor, such as Windows Notepad.
**M. To set up the drawing environment**

The scope of the section is determined by first setting the overall dimensions of the workspace to extend to (at least) the maximum dimensions of the section that is to be considered. A grid is then placed within this workspace, which if selected in the options, can be snapped to in order to accurately locate the coordinates of the boundary nodes.

1. Select either:
   
   Edit > Overall dimensions
   
   or
   
   the Overall dimensions tool
   
   The Overall dimensions dialog opens

2. Type the maximum width and height of your section.

   **Tip:** This can be edited later by repeating this procedure.

3. Click OK.
   
   The section axis is displayed.

4. To set up a grid, do the following:
   
   a. Select either:
       
       Settings > Grid options
       
       or
       
       the Grid step tool
       
       The Grid parameters dialog opens.
   
   b. Type the Y and Z direction grid increments.
   
   c. (Optional) Type an Angle used to orient the grid.
   
      The angle is measured positive counter-clockwise from the Y axis.
   
   d. Click OK.

5. To display the grid, select either:
   
   Settings > Grid
   
   or
   
   the Grid tool
   
   The grid is displayed and the tool is depressed on the toolbar and in the menu.

6. To snap points to the grid, select Settings > Snap to grid.

   **Note:** This tool is on by default when the grid is displayed.

**M. To draw an external contour**

Used to define the outside of the section shape.

Only one closed shape may be used to define a cross section.
1. Select either
   \textit{Edit > Draw External Contour}
   
or
   the \textit{External Contour} tool

2. Click any point in the View window as the first vertex of the exterior of the shape.
   A line is rubber-banded between this point and the mouse pointer.

3. Click subsequent points to form additional vertices.

   \textbf{Tip:} You may right-click to undo a previous vertex.

4. Double click the final point once you have formed a closed geometric shape.
   The final point does not need to be a previously entered data point. The rubber band to the first point will give you a preview of what the shape will look like for any point in the View window.
   The shape is shaded solid.

5. Exit the external contour drawing mode by either
   - repeating step one
   - or
   - select some other drawing mode

\textit{M. To draw an internal contour}

Used to define the openings or negative spaces within the cross section shape.
Inner contours must be closed shapes. Inner contour areas or vertices may not overlap.

1. Select either
   \textit{Edit > Draw Internal Contour}
   
or
   the \textit{Internal Contour} tool

2. Click any point in the View window as the first vertex of the inner contour.
   A line is rubber-banded between this point and the mouse pointer.

3. Click subsequent points to form additional vertices.

   \textbf{Tip:} You may right-click to undo a previous vertex.

4. Double click the final point once you have formed a closed geometric shape.
   The final point does not need to be a previously entered data point. The rubber band to the first point will give you a preview of what the shape will look like for any point in the View window.
   The shading inside of the shape is removed.

5. Exit the internal contour drawing mode by either
   - repeating step one
   - or
   - select some other drawing mode
**M. To round a corner**

Uses this procedure to round –also referred to as fillet– interior or exterior contour corners.

1. Select either
   
   **Edit > Radius Corner...**
   
   or
   
   the **Smooth tool**
   
   2. Select any interior or exterior contour vertex.

   **Tip:** When hovering over a vertex, the mouse pointer changes to a double-line cross-hair (\(\phi\)).

   The Smooth Radius dialog opens.

   3. Type a radius value, \(R\), and click **OK**.
   
   The previously specified radius is the default value.

   The corner is redrawn as a series of points forming an arc with the specified radius.

   4. Exit the radius drawing mode by either
   
   repeating step one
   
   or
   
   selecting some other drawing mode

**M. To insert a round opening**

Use the following procedure to insert a round hole by one of two means.

A hole is an interior contour which is parametrically defined with a center and radius. You may specify a radius by an exact value or by drawing in the View window, depending on your tool selection.

1. Select either
   
   the **Circle hole tool**
   
   or
   
   the **Circle hole with specified radius tool**
   
   If you selected to specify a radius, the **Radius of the circle hole** dialog opens.

   2. If you selected to specify a radius, enter the value and click **OK** in the **Radius of the circle hole** dialog.

   3. Click a point to define the center of the circular hole.
   
   If you are defining the radius graphically, a circle is rubber-banded to the mouse pointer. Otherwise, the hole is drawn with the previously specified radius.

   4. Double click a point to define a radius.
   
   The hole is drawn.

   5. Exit the hole drawing mode by either
   
   repeating your selection in step one
   
   or
selecting some other drawing mode

**M. To copy an internal contour**

Use this procedure to make a copy of any internal contour or hole.

Inner contours must be previously specified for the tools used to be active.

Inner contours can be placed anywhere within the external contour.

1. Select either

   - **Edit > Copy inner contour**
   
   or
   
   the **Copy inner contour(s)** tool

2. From there, select either to use a **Rectangle** or **Polygon** window method for performing a selection.

   Depending on the window method selected, the mouse pointer either changes to a rectangular selection icon (rectangular) or a polygonal selection icon (polygon).

3. Click an initial point outside of the inner contour(s) you wish to copy.

   Depending on the window method selected, either a rectangle (rectangular) or a single line (polygon) is rubber banded to your mouse pointer.

4. Select the next point based on the following:

   For a rectangle, select the opposite corner of the rectangular window enclosing all of the inner contours you wish to copy.

   or

   For a polygon, click the next vertex along the polygon defining the outer edge of the window area. Repeat this step until the contours you wish to copy are enclosed, double-clicking the last point to close the polygon.

5. Specify a location of the copy by either

   Clicking anywhere within the windowed area and dragging the copy to where you wish to place it. Your mouse pointer becomes a multi-arrow icon to indicate this option.

   or

   Right-click to open the **Copy of contour(s)** dialog which is used to specify Y and Z offsets. Click **OK** to place the copy at this location.

Copies inner contours or their vertices may not overlap with other inner contours.

**M. To delete vertices**

Use this procedure to remove the vertices of external or internal contours.

1. Select either

   - **Edit > Delete vertices**
   
   or

   the **Delete vertices** tool
2. From there, select one of the following methods for vertex selection:
   Single
   or
   Rectangle
   or
   Polygon
   The mouse pointer updates to reflect the selection method.
3. Select either:
   a single vertex to delete; the mouse pointer changes to a double-line cross-hair (▲)
   or
   one corner of a rectangle window; the mouse pointer changes to a rectangular selection icon (□)
   or
   the first vertex of a polygon window; the mouse pointer changes to a polygonal selection icon (■)
   A single point selection is deleted and the shape is redrawn connecting the adjacent points. Once one or more vertices are enclosed in a completed rectangle or polygon window area, then those vertices are deleted and the adjacent vertices to those are connected.

M. To delete an opening

Use this procedure to delete an interior contour or hole from the shape.
Inner contours must be previously specified for the tools used to be active.

1. Select either
   Edit > Delete internal contour
   or
   the Delete tool
2. Click anywhere inside the contour or hole you want to remove.
   The internal contour is removed from the shape.
3. Exit the delete contour mode by either:
   repeating step one
   or
   selecting some other drawing mode

M. To shift the coordinate center

Use this procedure to shift the coordinate center to any arbitrary point or to the center of gravity.

1. Select Edit > Coordinate Center
   The Coordinate Center dialog opens.
2. Specify a new coordinate center by either
    specifying both Y and Z shift values and click Apply
    or
    click Shift to specify the current shape center of gravity.

    **Note:** The origin must remain within the overall dimensions of the workspace.

    The coordinate origin shifts to the specified location.

**M. To import a CAD drawing**

This procedure is used to import a 2D CAD, closed area as a section profile. This allows you to use the CAD program of your choice to generate complex cross sections.

The following AutoCAD® entity types are supported in the import facility:

- 3DFACE
- SOLID
- TRACE
- LINE
- POLYLINE
- LWPOLYLINE
- ELLIPSE
- CIRCLE
- ARC

1. Select **File > Import DXF**
   The Import dialog opens.
2. Navigate to and select the .dxf file which you want to import.

    **Note:** The files that are to be imported are such that all the vertices are in a single plane and form closed areas.

3. Click **Open**.

   **To export shape for use in STAAD.Pro**

To export the shape profile for use in STAAD.Pro as a user-provided table, do the following.

    **Note:** This profile shape should consist of only an outer contour. Any internal voids such as internal contours or openings will be ignored, though section properties calculated by Section Wizard based on sections with voids will be used.

1. Select **File > Export to STAAD.Pro > General Section.**
   A **Save As** dialog opens.
2. Navigate to where you want to save the file, type a file name, and then click **Save**.
   The **User Table Units** dialog opens.
3. Select the unit of length for use with this shape and then click **OK**.
   The **General** shape dialog opens.
4. Type the **Section Name** for use in the STAAD.Pro user provided table.
5. (Optional) Type in any override values for section properties you want to use instead of those calculated by Section Wizard.

The shape vertices and calculated section properties are saved to the .upt file.

You can use this file as an external file for a user-provided section. Refer to To use a general shape created in Section Wizard (on page 739).

**M. Application Window Layout**

**Drawing Window**

Left click selects a data point. Double click to accept. Right click to void last data point or current action (same as selecting the Cancel operation tool).

**M. Toolbar**

Contains several of the most commonly used tools found in the Free Sketch module.

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Same effect as selecting …</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Opens an empty section design file. If the currently loaded section has been modified, then the option is given to save it prior to commencing with the new design.</td>
<td>File &gt; New</td>
</tr>
<tr>
<td>Open</td>
<td>Opens the Open Data file dialog, which is used to select an existing Section Sketch Data file (file extension .cns). Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the Preview option is checked, then a preview of the section can be displayed prior to opening the file.</td>
<td>File &gt; Open …</td>
</tr>
<tr>
<td>Parametric Section</td>
<td>Opens the Parametric Section dialog which is used to quickly create sections based on a section type and a number of key dimensions and insert that section as an external contour.</td>
<td>File &gt; Parametric Sections …</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>----------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>-----------------------------</td>
</tr>
<tr>
<td><img src="image" alt="Save" /></td>
<td>Saves any changes made to the current data file. <strong>Note:</strong> If the current file has not been previously saved, this has the same effect as selecting <strong>Save As</strong>…</td>
<td>File &gt; Save</td>
</tr>
<tr>
<td><img src="image" alt="Find Equivalent Section" /></td>
<td>Opens the <strong>Equivalent Section module</strong> (on page 757) with the current section properties as input values, which can be used to search for an Equivalent section to the current drawing.</td>
<td>File &gt; Equivalent Section …</td>
</tr>
<tr>
<td><img src="image" alt="Section Builder" /></td>
<td>Launches the <strong>Section Builder module</strong> (on page 744).</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Undo" /></td>
<td>Negates the last drawing operation. You can undo a series of modifications by repeatedly selecting Undo.</td>
<td>Edit &gt; Undo</td>
</tr>
<tr>
<td><img src="image" alt="Redo" /></td>
<td>Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting Redo.</td>
<td>Edit &gt; Redo</td>
</tr>
<tr>
<td><img src="image" alt="Cancel Operation" /></td>
<td>Cancels the current drawing operation.</td>
<td>ESC</td>
</tr>
<tr>
<td><img src="image" alt="Overall Dimensions" /></td>
<td>Opens the <strong>Overall Dimensions dialog</strong>, which is used to specify the size of the workspace that will be required to create the section. The minimum it should be set to is the overall dimensions of the section but can be set larger.</td>
<td>Edit &gt; Overall Dimensions …</td>
</tr>
<tr>
<td><img src="image" alt="Grid Setup" /></td>
<td>Opens the <strong>Grid Parameters dialog</strong>, which is used to specify the grid spacing along the Y and Z directions and the Angle of rotation of the grid axes.</td>
<td>Settings &gt; Grid Options …</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>-----------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>-------------------------------------------</td>
</tr>
<tr>
<td><img src="78x655" alt="Image" /></td>
<td>Used to set or clear the external contour drawing mode, which defines the outside of the section.</td>
<td>Edit &gt; Draw External Contour</td>
</tr>
<tr>
<td>External Contour</td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="78x597" alt="Image" /></td>
<td>Used to set or clear the internal contour mode, which is used for creating internal openings within an external boundary.</td>
<td>Edit &gt; Draw Internal Contour</td>
</tr>
<tr>
<td>Internal Contour</td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="78x538" alt="Image" /></td>
<td>Used to set or clear the circular hole drawing mode, which is used to define the center of a hole (e.g., a parametrically defined, circular inner contour) and radius graphically.</td>
<td>Edit &gt; Insert circular hole, center and radius</td>
</tr>
<tr>
<td>Circle hole</td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="78x455" alt="Image" /></td>
<td>Insert circular hole with defined radius</td>
<td>Edit &gt; Insert circular hole with defined radius</td>
</tr>
<tr>
<td>Circle hole with specified radius</td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="78x397" alt="Image" /></td>
<td>Used to window either a rectangle or polygonal area to select one or more internal contours for copying. Copies can be placed manually or by specifying offset distances.</td>
<td>Edit &gt; Copy inner contour</td>
</tr>
<tr>
<td>Copy inner contour(s)</td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="78x326" alt="Image" /></td>
<td>Used to window either a rectangle or polygon area to select one or more internal contours making multiple copying. Copies can be placed manually or by specifying offset distances and a number of copies.</td>
<td>Edit &gt; Multiple copy of inner contour</td>
</tr>
<tr>
<td>Multiple copies of inner contour(s)</td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="78x231" alt="Image" /></td>
<td>Used to set or clear the smooth corner drawing mode, which is used to select a corner to fillet and then opens the Smooth radius dialog to specify a radius value for the rounded corner.</td>
<td>Edit &gt; Radius Corner…</td>
</tr>
<tr>
<td>Smooth</td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="78x148" alt="Image" /></td>
<td>Used to set or clear the delete internal contour drawing mode, which is used to remove internal contours by clicking anywhere in their boundary.</td>
<td>Edit &gt; Delete internal contour</td>
</tr>
<tr>
<td>Delete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>--------------------</td>
<td>-------------------------------------------------------------------------------------------------------</td>
<td>------------------------------</td>
</tr>
<tr>
<td>Move Vertices</td>
<td>Used to window either a rectangular or polygonal area to select one or more vertices to move. Vertices can be moved either manually or by specifying offset distances.</td>
<td>Edit &gt; Move vertices</td>
</tr>
<tr>
<td>Delete Vertices</td>
<td>Used to select a single vertex or window either a rectangular or polygonal area to select one or more vertices for deletion.</td>
<td>Edit &gt; Delete vertices</td>
</tr>
<tr>
<td>Edit coordinates of Vertices</td>
<td>Displays all vertex numbers in the View window and opens the Coordinates dialog which is used to specify precise coordinates of each vertex for a selected contour.</td>
<td>Edit &gt; Edit vertices</td>
</tr>
<tr>
<td>Calculation</td>
<td>First opens the Poisson's Ratio dialog which is used to specify a value for ( \nu ). Calculates the cross-sectional properties of the composite section and the opens the Basic Geometry dialog to display the values.</td>
<td>File &gt; Calculation</td>
</tr>
<tr>
<td>Report</td>
<td>Formats and sends the data to the Report Type option defined in the Misc sheet of the Program Options dialog. See Options below.</td>
<td>File &gt; Report</td>
</tr>
<tr>
<td>Stress Field</td>
<td>Opens the Section forces dialog, which is used to specify major axis moments and axial force on the section. Click OK to close the dialog and display the stress distribution on the section drawing.</td>
<td>File &gt; Stress field</td>
</tr>
<tr>
<td>Options</td>
<td>Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.</td>
<td>Settings &gt; Preferences …</td>
</tr>
<tr>
<td>Grid</td>
<td>Used to set or clear the display of the drawing grid in the View window.</td>
<td>Settings &gt; Grid</td>
</tr>
<tr>
<td>Tool</td>
<td>Description</td>
<td>Same effect as selecting …</td>
</tr>
<tr>
<td>-------------------------</td>
<td>------------------------------------------------------------------------------</td>
<td>------------------------------------------</td>
</tr>
<tr>
<td>![Center of Gravity]</td>
<td>Used to set or clear the icon that shows the location of the center of gravity.</td>
<td>Settings &gt; Center of Gravity</td>
</tr>
<tr>
<td>![Shear Center]</td>
<td>Used to set or clear the icon that shows the location of the shear center.</td>
<td>Settings &gt; Shear Center</td>
</tr>
<tr>
<td>![Show Coordinate Axis]</td>
<td>Used to set or clear the display of the coordinate axis.</td>
<td>Settings &gt; Coordinate Axis</td>
</tr>
<tr>
<td>![Principle Axes]</td>
<td>Used to set or clear the display of the principle axis of the current cross section.</td>
<td>Settings &gt; Principle Axis</td>
</tr>
<tr>
<td>![Zoom In]</td>
<td>Used to increase the magnification in the View window.</td>
<td>Settings &gt; Zoom In</td>
</tr>
<tr>
<td>![Zoom Out]</td>
<td>Used to decrease the magnification in the View window.</td>
<td>Settings &gt; Zoom Out</td>
</tr>
<tr>
<td>![Help]</td>
<td>Opens the Section Builder help window.</td>
<td>Help &gt; Help Topics</td>
</tr>
<tr>
<td>![About]</td>
<td>Opens the About Section Builder window, which displays version and copyright information.</td>
<td>Help &gt; About ...</td>
</tr>
</tbody>
</table>

**M. Menus**

Menus in the Section Builder module window.

**M. File menu**

Contains items for creating, opening, and closing section data files, printing, and exiting the program.
### Table 37: File menu items

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Opens an empty section design file. If the currently loaded section has been modified, then the option is given to save it prior to commencing with the new design.</td>
<td>CTRL+N</td>
</tr>
<tr>
<td>Open</td>
<td>Opens the Open Data file dialog, which is used to select an existing Section Sketch Data file (file extension .cns). Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the Preview option is checked, then a preview of the section can be displayed prior to opening the file.</td>
<td>&lt;CTRL+O&gt;</td>
</tr>
<tr>
<td>Save</td>
<td>Saves any changes made to the current data file. <strong>Note:</strong> If the current file has not been previously saved, this has the same effect as selecting Save As…</td>
<td>&lt;CTRL+S&gt;</td>
</tr>
<tr>
<td>Save As...</td>
<td>Opens the Save data file dialog which is used to specify a file name and location for the section data file. Files are saved in the Free Sketch (.CNS) format.</td>
<td></td>
</tr>
<tr>
<td>Close</td>
<td>This closes the current section and leaves Free Sketch ready to start a new file or open an existing file.</td>
<td></td>
</tr>
<tr>
<td>Report</td>
<td>Formats and sends the data to the Report Type option defined in the Misc sheet of the Program Options dialog. See Options below.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
<td>Same effect as selecting…</td>
</tr>
<tr>
<td>--------------------</td>
<td>------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Calculation</td>
<td>First opens the Poisson's Ratio dialog which is used to specify a value for Nu. Calculates the cross-sectional properties of the composite section and the opens the Basic Geometry dialog to display the values.</td>
<td></td>
</tr>
<tr>
<td>Stress Field</td>
<td>Opens the Section forces dialog, which is used to specify major axis moments and axial force on the section. Click OK to close the dialog and display the stress distribution on the section drawing.</td>
<td></td>
</tr>
<tr>
<td>Parametric Sections...</td>
<td>Opens the Parametric Section dialog which is used to quickly create sections based on a section type and a number of key dimensions and insert that section as an external contour.</td>
<td></td>
</tr>
<tr>
<td>Rolled Section...</td>
<td>Opens the Rolled Section dialog, which is used to select a catalog section to insert into an new drawing file. The section can then be modified, just as you would with any other sketch.</td>
<td></td>
</tr>
<tr>
<td>Equivalent Section...</td>
<td>Opens the Equivalent Section module (on page 757) with the current section properties as input values, which can be used to search for an Equivalent section to the current drawing.</td>
<td></td>
</tr>
<tr>
<td>Export to STAAD.Pro &gt;</td>
<td>Select either to export the section as a General or Prismatic section. A Save If an existing User Provide Table file is selected, the current section will be appended to the end of this table. Refer to the STAAD.Pro Technical Reference for additional information on using section types.</td>
<td></td>
</tr>
</tbody>
</table>
### Menu item

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Import DXF</td>
<td>Opens a File Open dialog which is used to select CAD files to import in the *.DXF file format</td>
<td></td>
</tr>
<tr>
<td>Send…</td>
<td>Opens an e-mail (using your default e-mail client) and attaches the current data file.</td>
<td></td>
</tr>
<tr>
<td>Recent File list</td>
<td>A list of the most recent four section data files. Select one to open that data file.</td>
<td></td>
</tr>
<tr>
<td>Exit</td>
<td>Closes the program. If any changes were made since the last time the data file was saved, a message dialog opens confirming you wish to save those changes.</td>
<td><code>&lt;ALT+F4&gt;</code></td>
</tr>
</tbody>
</table>

### M. Parametric Sections dialog

Used to quickly create sections based on a section type and a number of key dimensions and insert that section as an external contour.

Opens when File > Parametric Sections … is selected.

**Dialog controls**

- **section type selector**

  Select the general section type that you wish to use from the graphical list. The list contains 22 different generic section types. The parameters and diagram update according to the selection.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Section Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Symbol 1]</td>
<td>Angle</td>
</tr>
<tr>
<td>![Symbol 2]</td>
<td>Rectangle</td>
</tr>
<tr>
<td>![Symbol 3]</td>
<td>Rounded Rectangle</td>
</tr>
<tr>
<td>![Symbol 4]</td>
<td>Triangle</td>
</tr>
<tr>
<td>![Symbol 5]</td>
<td>Six Side Polygon (Hexagon)</td>
</tr>
<tr>
<td>![Symbol 6]</td>
<td>Eight Side Polygon (Octagon)</td>
</tr>
<tr>
<td>Symbol</td>
<td>Section Type</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------</td>
</tr>
<tr>
<td>7</td>
<td>Rectangular Hollow Section</td>
</tr>
<tr>
<td>8</td>
<td>Rounded Rectangular Hollow Section</td>
</tr>
<tr>
<td>9</td>
<td>Channel</td>
</tr>
<tr>
<td>10</td>
<td>Rounded Channel</td>
</tr>
<tr>
<td>11</td>
<td>Flanged Rectangular Hollow Section</td>
</tr>
<tr>
<td>12</td>
<td>Unsymmetric I Section</td>
</tr>
<tr>
<td>13</td>
<td>Rounded Symmetric I Section</td>
</tr>
<tr>
<td>14</td>
<td>Rounded Tee Section</td>
</tr>
<tr>
<td>15</td>
<td>Double Tee Section</td>
</tr>
<tr>
<td>16</td>
<td>Wedge</td>
</tr>
<tr>
<td>17</td>
<td>Z Section</td>
</tr>
<tr>
<td>18</td>
<td>Cruciform</td>
</tr>
<tr>
<td>19</td>
<td>Circle</td>
</tr>
<tr>
<td>20</td>
<td>Hollow Pipe</td>
</tr>
<tr>
<td>21</td>
<td>Half Pipe</td>
</tr>
<tr>
<td>22</td>
<td>Ellipse</td>
</tr>
</tbody>
</table>
section parameters Fields for each of the section dimensions for the selected section type are displayed. Enter a dimension into each field (each section has at least one required dimension). Refer to the diagram for the selected section type for an explanation of each field.

OK Closes the dialog and adds an external contour with defined parametric section.

Cancel Closes the dialog and returns to the previous sketch file.

M. Rolled Section dialog
Used to select a catalog section to insert into a new drawing file. The section can then be modified, just as you would with any other sketch.

Opens when File > Rolled Sections ... is selected.

Dialog controls

catalog lists Catalog sections are listed in a tree format by Country/Standard, shape type, and section size. Expand the tree to locate the section you wish to add and select its entry.

OK Closes the dialog and adds an external contour with selected catalog section.

Cancel Closes the dialog and returns to the previous sketch file.

M. Edit menu
Contains items for

Table 38: Edit menu items

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Same effect as selecting…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo</td>
<td>Negates the last drawing operation. You can undo a series of modifications by repeatedly selecting Undo.</td>
<td>CTRL+Z</td>
</tr>
<tr>
<td>Redo</td>
<td>Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting Redo.</td>
<td>CTRL+Y</td>
</tr>
<tr>
<td>Overall Dimensions ...</td>
<td>Opens the Overall Dimensions dialog, which is used to specify the size of the workspace that will be required to create the section. The minimum it should be set to is the overall dimensions of the section but can be set larger.</td>
<td></td>
</tr>
<tr>
<td>Draw external contour</td>
<td>Used to set or clear the external contour drawing mode, which defines the outside of the section.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>Description</td>
<td>Same effect as selecting…</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
<td>--------------------------</td>
</tr>
<tr>
<td>Locate circular external contour</td>
<td>Opens the Radius of the external contour dialog, which is used to specify a radius value for a parametrically defined, circular external contour. <strong>Note:</strong> This menu item is only active if no other external contour has been defined.</td>
<td></td>
</tr>
<tr>
<td>Draw internal contour</td>
<td>Used to set or clear the internal contour mode, which is used for creating internal openings within an external boundary.</td>
<td></td>
</tr>
<tr>
<td>Copy inner contour &gt; &lt;list&gt;</td>
<td>Used to window either a rectangle or polygonal area to select one or more internal contours for copying. Copies can be placed manually or by specifying offset distances.</td>
<td></td>
</tr>
<tr>
<td>Multiple copy of inner contour &gt; &lt;list&gt;</td>
<td>Used to window either a rectangle or polygonal area to select one or more internal contours making multiple copying. Copies can be placed manually or by specifying offset distances and a number of copies.</td>
<td></td>
</tr>
<tr>
<td>Delete internal contour</td>
<td>Used to set or clear the delete internal contour drawing mode, which is used to remove internal contours by clicking anywhere in their boundary.</td>
<td></td>
</tr>
<tr>
<td>Insert circular hole, center, and radius</td>
<td>Used to set or clear the circular hole drawing mode, which is used to define the center of a hole (e.g., a parametrically defined, circular inner contour) and radius graphically.</td>
<td></td>
</tr>
<tr>
<td>Insert circular hole with defined radius</td>
<td>Used to set or clear the circular hole with defined radius drawing mode, which is used to define the center of a hole graphically and then specify an exact radius.</td>
<td></td>
</tr>
</tbody>
</table>
### Menu item
### Description
### Same effect as selecting...

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Parametric hole</td>
<td>Opens the Parametric holes dialog, which is used to define the location and dimensions of either a circular or rectangular inner contour.</td>
<td></td>
</tr>
<tr>
<td>Radius corner</td>
<td>Used to set or clear the smooth corner drawing mode, which is used to select a corner to fillet and then opens the Smooth radius dialog to specify a radius value for the rounded corner.</td>
<td></td>
</tr>
<tr>
<td>Move vertices</td>
<td>Used to window either a rectangular or polygonal area to select one or more vertices to move. Vertices can be moved either manually or by specifying offset distances.</td>
<td></td>
</tr>
<tr>
<td>Edit vertices</td>
<td>Displays all vertex numbers in the View window and opens the Coordinates dialog which is used to specify precise coordinates of each vertex for a selected contour.</td>
<td></td>
</tr>
<tr>
<td>Delete vertices</td>
<td>Used to select a single vertex or window either a rectangular or polygonal area to select one or more vertices for deletion.</td>
<td></td>
</tr>
<tr>
<td>Coordinate center</td>
<td>Opens the Coordinate Center dialog, which is used to specify shift the global coordinates to either an arbitrary location or to the center of gravity of the shape.</td>
<td></td>
</tr>
</tbody>
</table>

**M. Settings menu**

Contains items for controlling non-element specific settings as well as the behavior and display of the drawing environment.

**Table 39: Settings menu items**

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Preferences</td>
<td>Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.</td>
<td></td>
</tr>
</tbody>
</table>

M. Properties and Specifications
### Menu item | Description | Shortcut
---|---|---
Grid Options ... | Opens the Grid Parameters dialog, which is used to specify the grid spacing along the Y and Z directions and the Angle of rotation of the grid axes. |  
Grid | Used to set or clear the display of the drawing grid in the View window. |  
Snap to grid | Used to set or clear the snap to grid behavior, which provides drawing assistance by selecting the nearest grid point to a selected point in the View window. |  
Snap to vertices | Used to set or clear the snap to vertices behavior, which provides drawing assistance by selecting the nearest vertex point to selected in the View window. |  
Center of Gravity | Used to set or clear the icon that shows the location of the center of gravity. |  
Shear Center | Used to set or clear the icon that shows the location of the shear center. |  
Coordinate Axis | Used to set or clear the display of the coordinate axis. |  
Principle Axis | Used to set or clear the display of the principle axis of the current cross section. |  
Zoom In | Used to increase the magnification in the View window. |  
Zoom Out | Used to decrease the magnification in the View window. |  

M. Preferences dialog  
Used to set the units, colors, output types, and databases for standard sections. 
Opens when **Settings > Preferences ...** is selected.  
Units of measurement tab
Select the units for angles, section dimensions, section properties, axial forces and moments, as well as the number of decimal places and whether or not to use an exponential form.

Misc. tab

Select the language, Report settings such as the template document, whether the documents should use Word 7 or Word 97 file formats, the paper size and font. Set whether documents should be sent direct to the printer or into Word for editing. Additional options determine the number of vertices to created by a circle, option to snap to grid and whether the ends of vertices are indicated by a circle or not.

Stress scale tab

Used to set the colors of the maximum compression and tension stresses and the colour these grade to for zero stress, along with the number of bands to be displayed.

Profile databases tab

M. Grid Parameters dialog
Used to specify the grid spacing along the Y and Z directions and the Angle of rotation of the grid axes.

Opens when Settings > Grid Options ... is selected.

Dialog controls

Y       Grid spacing along the section local y axis (horizontal on screen).
Z       Grid spacing along the section local z axis (vertical on screen).
Angle   Angle from the coordinate axis to the grid lines, taken counterclockwise. Accepted values range between zero (inclusive) and 90 degrees. Negative values are not accepted. Only the drawing grid is rotated; no section data is changed.
OK      Saves changes and closes the dialog.
Cancel  Closes the dialog without saving any changes.

M. Service menu
Contains items for opening tools and utilities related to the Free Sketch module.

Table 40: Service menu items

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows Calculator ...</td>
<td>Opens the Windows system Calculator window.</td>
</tr>
<tr>
<td>Formula Calculator ...</td>
<td>Opens the Formula Calculation window (on page 784) which is used to evaluate a formula, from the numerical result can then be copied back into the Free Sketch module.</td>
</tr>
<tr>
<td>Unit Converter ...</td>
<td>Opens the Unit Converter Window which is used to as a value converter between different preset units of measurement. Select the tab pertaining to the type of measurement and then enter the known value in the appropriate field. Upon pressing &lt;Enter&gt;, all the other fields are updated to equivalent values.</td>
</tr>
</tbody>
</table>
Menu item | Description
--- | ---
Section Builder ... | Launches the Section Builder module (on page 744).

**M. Help menu**
The Help menu contains items for using online help.

**Table 41: Help menu items**

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Help Topics</td>
<td>Opens the Section Builder help window.</td>
<td>F1</td>
</tr>
<tr>
<td>About</td>
<td>Opens the About Section Builder window, which displays version and copyright information.</td>
<td></td>
</tr>
</tbody>
</table>

**M. Formula Calculation window**
Used for holding evaluations under the formulas, which are set by the user in the window of lead

![Formula calculation window](image)

*Figure 112: Formula calculation window*

**Formula Syntax**
At lead of the formulas it is necessary to observe the following rules:
- the names of functions are entered by lower case letters of the Latin alphabet;
- a separator of fractional and whole parts of number is the point;
- the arithmetical operations are set by characters +, -, *, /, exponentiation ^ (for example, 2.5*2.5*2.5 is typed as 2.5 ^ 3).

**Supported Functions**
At record of the formulas it is possible to use the following functions:
floor - the greatest integer not exceeding preset
tan - a tangent
sin - sine
cos - cosine
asin - an arcsine
acos - an arccosine
atan - an arctangent
exp - an exponential curve
ceil - the least integer exceeding preset
tanh - a tangent hyperbolic
sinh - sine hyperbolic
cosh - cosine hyperbolic
log - a Napierian logarithm
log10 - a Brigg's logarithm
abs - an absolute value
sqrt - the radical square

Depending on a state of the switch Degrees / radians, arguments trigonometrically functions (sin, cos, tan) and
the outcomes return trigonometrically functions (asin, acos, atan) are reduced in degrees or radians accordingly.

Usage only of parentheses is admitted at arbitrary depth of an enclosure.

**Examples**

The formula:

\[ 1.2 + \sin (0.43) + 6.7\sqrt{6.8 - \sqrt{\sqrt{0.003}}} \]

is typed as follows:

\[ 1.2 + \sin (0.43) + 6.7\sqrt{6.8 - \sqrt{\sqrt{0.003}}} \]

Click Variable and then the \(x\), \(y\), and \(z\) fields become active. Thus values of variables are set in
appropriate windows of lead. It allows to carry out a series of one-type evaluations at different values
of parameters. For example, in this condition the following formula

\[ 1.2 + \sin (x) + 6.7\sqrt{6.8 - \sqrt{y}} \]

is typed as follows:

\[ 1.2 + \sin (x) + 6.7\sqrt{6.8 - \sqrt{y}} \]

Then click Calculate to have the program resolve the variables into the formula.

Click Copy to copy the result to the Windows clipboard.

Moreover, the program allows to input some symbolic expression (depending on the variables \(x,y,z\));

\[ \frac{\partial f}{\partial x}, \frac{\partial f}{\partial y}, \text{ or } \frac{\partial f}{\partial z} \]

and retrieve symbolic expression for the corresponding partial derivative.

**M. Unit Converter**

This is a utility program used to convert measurements from a known units of measurement into several
different units.
The dialog includes several tabs, each corresponding to a type of measurement.

- Length
- Area
- Volume
- Force
- Angle
- Pressure
- Moment of Couple

To convert units

1. Select the tab with the type of measurement you want to convert.
2. Type the known value in the field with the known units.
3. Press <Enter>. The value is converted in each field with different units on this tab.

Tip: You can double-click in a field to quickly select the contents. Right-click and then select Copy from the pop-up menu to save this value to the Windows clipboard.

Click Exit when you are finished converting values.

M. Section Database Manager

This utility application is used to manage the section catalog databases used in STAAD.Pro.

M. To change a default section database

To change the section database used the default for a particular country and material combination, use the following procedure.

1. Select the Section Database tool in the Tools group on the Specifications ribbon tab.

The Section Database Manager application opens.

2. On the Home ribbon tab, select the Configuration tool in the Options group.

The Default Database Configuration dialog opens.

3. (Optional) Select the Default Country from the drop-down list.
   This will be the first country expanded when the Section Profiles Tables dialog open is opened.

4. Select the tab corresponding to the material type.

5. For a particular Country, select a default database from the drop-down list in the Selected Database field.

6. Repeat steps 4 and 5 to change additional default databases.

7. Click Save.

8. Click Close.
Alternatively, you can navigate to the database you want to use as the default for a particular country and material combination in the Contents panel. Then, right-click and select Set as Default Database from the pop-up menu.

M. To add a custom table section property

To edit the section database tables to include custom sections, use the following procedure.

Note: This is not the same as adding a user-defined section or catalog. This should only be performed if you need to make edits to the standard shape catalogs that are included with STAAD.Pro.

1. Select the Section Database tool in the Tools group on the Specifications ribbon tab.

The Section Database Manager application opens.

2. In the Databases list, double-click on the country, material, catalog, and table you want to edit.

   The selected table opens in read-only mode.

3. Select the Lock tool.

   The table is unlocked for editing.

4. Either:

   To... Do the following...

   add a row to the end of the table select the Append Row tool

   add a row above the current table row select the Add Above tool

   add a row below the current table row select the Add Below tool

   The order of the table here is how it will also be presented in the Section Profile Tables dialog in STAAD.Pro.

   An empty row is added to the table.

5. Type a record number (RECNO), name, STAAD.Pro nomenclature name (StaadName), and then the section values for each column in the table.

   You can press <Tab> to move to the next column within a row when entering values.

6. (Optional) Repeat steps 4 and 5 as many times as needed to add additional shapes.

7. Click Commit to save the changes to the database.

8. Select the Unlock tool.

   The database is now locked against further editing.

9. Close the SectionDBManager program.

You can now use this section just as you would any default catalog section in STAAD.Pro.

Note: You can use the Delete Row tool to remove the current selected table row. Exercise caution when using this tool!
**Default Database Configuration** dialog

Used to select the default country and section databases to use in the **Section Profiles Table** dialog.

Opens when the **Configuration** tool is selected on the **Home** tab in the **Section Database Manager** application.

When an existing STAAD.Pro model is opened, the program reads the contents of the file, and checks the validity of data in that file. One of those data items validated is names of sections assigned from steel tables. Since steel sections are country-specific, such as British, German, etc., the program needs to know the country or organization whose steel table is the underlying database for validating the sections being read in from the file. Normally, the input file contains the name of the database as part of the member property command. In the absence of an explicit name, STAAD.Pro uses a default. That default is set using this facility.

**Section Profile Table tab**

**Default Country**

Select the default country to display when the **Section Profiles Table** dialog opens.

**Tables Selection**

The materials are organized as tabs along the top of the table. For each material, the countries are listed along with the default catalog to use for each. Select the catalog from the drop-down list for each as needed.

**Custom Profile Table**

Used to associate a user-defined section table with STAAD.Pro.

STAAD.Pro profile table files are Microsoft Office Access® database files (file extension .mdb).

**Category**

Select the profile category (typically material and/or country) with which the profile database is to be associated.

**Table**

Type the name of the STAAD.Pro profile table file (file extension .mdb) to add or click [...] to browse for a file.

**Add**

Adds the selected custom profile table to the program.

### M. Materials and Constants

You can use material definitions or assign individual material constant values to beams, plates, and solids.

**Note:** Material definitions are recommended for most models.

### M. To create a material definition

To create a new isotropic material definition for use with your model, use the following procedure.

You may want to set the input units to a familiar set of units for defining materials before creating a new material definition.

STAAD.Pro includes a set of predefined materials for concrete, aluminum, steel, and stainless steel.


2. In the Material - Whole Structure dialog Isotropic tab, click Create.

   The Isotropic Material dialog opens.
3. Type a **Title** to identify the material.
   This title must be different than the predefined material names or any other existing user-defined material name.

4. Type the values of the Material Properties used for analysis:
   
   - Young’s Modulus, $E$
   - Poisson’s Ratio, $\nu$
   - Density, $\gamma$
   - Thermal Coefficient, $\alpha$
   - Critical Damping
   - Shear Modulus, $G$

5. Select the **Type of Material** from the drop-down list.

6. Type the strength values in the **Design Properties** section.
   
   The fields corresponding to the selected material type are active.

7. Click **OK**.

The material definition is now added to the **Materials** table and is available for selecting when assigning shapes and in **Material - Whole Structure** dialog.

**Related Links**

- *TR.26.1 Define Material* (on page 2501)
M. To create an orthotropic material

To create a 2D orthotropic material definition for use with orthotropic plate elements, use the following procedure.

You may want to set the input units to a familiar set of units for defining materials before creating a new material definition.

**Note:** 2D orthotropic material definitions are for use with plate elements only.

1. Select the **Materials** page in the **Analytical Modeling** page control bar.

   The **Material - Whole Structure** dialog and the **Materials** table open.

2. In the **Material - Whole Structure** dialog, select the **Orthotropic 2D** tab.

3. Click **Create**.
The 2-D Orthotropic Material Property dialog opens.

4. Type a Title for the material.
5. Type the material values:
   a. Type the Young's Modulus and Thermal Coefficients in the X direction.
   b. Type the Young's Modulus and Thermal Coefficients in the Y direction.
   c. Type the Density, Critical Damping, and Poisson's Ratio values.
      The same values are used in both X and Y directions.
   d. Type the Shear Modulus values for in-plane shear ($G_{xy}$), shear transverse to the local Y-Z direction ($G_{yz}$), and shear transverse to the local Z-X direction ($G_{zx}$).
6. Click Add.

The material definition is now added to the Materials table and is available for selecting when assigning shapes and in Material - Whole Structure dialog.

Related Links
- TR.26.1 Define Material (on page 2501)
- 2D Orthotropic Material Property dialog (on page 2989)

M. To assign material definitions

To assign material definitions to model objects (beams, plates, or solids), use the following procedure.

Often, you will assign a material definition to a beam along with the section assignment or to a plate with the thickness definition, as the material and profile or thickness are typically intrinsically associated. However, if you need to change materials or make assignments separately, this is done using the Material - Whole Structures dialog.


2. Select the Materials page on the Analytical Modeling workflow page control bar.

3. Assign the materials using one of the standard assign methods in the Material - Whole Structures dialog.

M. To assign material constants

To assign material constants to model objects (beams, plates, or solids), use the following procedure.

You may want to set the input units to a familiar set of units for a particular material value before assigning a material constant.

Tip: It is recommended to use material definitions in place of material constants. Material definitions also allow you to assign design strength properties.

1. Select the model object which have the same material constant.
2. On the Specifications ribbon tab, select the Constants tool in the Materials group.

A drop-down list of material constants opens.

3. Select the constant you want to assign:

   - Young's Modulus
   - Poisson's Ratio
   - Shear Modulus
   - Density
   - Thermal Coefficient
   - Damping Ratio

The corresponding dialog for the material constant opens.

4. Either:
   - select a predefined material constant value for a built-in material name: Aluminum, Concrete, or Steel
   - select the Enter Value option and type a value for the material constant

5. Select the To Selection option to limit the assignment to the selection set.

6. Click OK.

Related Links
- Material Constant dialog (on page 2977)
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)

M. Member Orientation

Related Links
- M. To change a beam incidence (on page 892)

M. To assign a member rotation angle

To assign an arbitrary member rotation angle about its longitudinal axis, use the following procedure. The rotation of a member about it's longitudinal axis is referred to as the "beta angle" in STAAD.Pro.

1. Select one or more members in the view window.
2. On the Specification ribbon tab, select the Beam > Beta Angle tool in the Specifications group.

The Beta Angle dialog opens.

3. Select the Angle in Degrees option and then type the angle of rotation.
4. Select the To Selection option.
M. To align a single angle to its flanges

To orient a single angle member aligned to its flanges, use the following procedure.

1. Select one or more members with single angle profiles in the view window.
2. On the Specification ribbon tab, select the Beam > Beta Angle tool in the Specifications group.

The Beta Angle dialog opens.

3. Select the option to align the flanges (i.e., geometric axis):

<table>
<thead>
<tr>
<th>To align...</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>the long leg with the local y axis (ST angles)</td>
<td>select the Angle option. This will rotate the angle section by $90^\circ - \alpha$.</td>
</tr>
<tr>
<td>the short leg with the local y axis (ST angles)</td>
<td>select the RAngle option. This will rotate the angle section by $180^\circ - \alpha$.</td>
</tr>
</tbody>
</table>

where

$$\alpha = \text{the angle between the principle axis system and the geometric axis (i.e., parallel to the flange faces) system of the single-angle profile}$$

Note: The orientation with the local y axis is for ST angles. RA angles are defined as being rotated $90^\circ$ and thus the Angle and RAngle options similarly rotate from that initial orientation.

4. Select the To Selection option.

Tip: The To View option assigns the specification to all members in the view window.

5. Click OK.
Table 42: Effect of BETA ANGLE and BETA RANGLE commands

<table>
<thead>
<tr>
<th>BETA value =</th>
<th>Zero (0)</th>
<th>ANGLE</th>
<th>RANGLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST Angle</td>
<td><img src="image1" alt="Diagram" /></td>
<td><img src="image2" alt="Diagram" /></td>
<td><img src="image3" alt="Diagram" /></td>
</tr>
<tr>
<td>RA Angle</td>
<td><img src="image4" alt="Diagram" /></td>
<td><img src="image5" alt="Diagram" /></td>
<td><img src="image6" alt="Diagram" /></td>
</tr>
</tbody>
</table>

Related Links

- G.4.3 Relationship Between Global and Local Coordinates (on page 2301)
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)
- Beta Angle dialog (on page 2978)

M. To align a member to a reference point

To orient a member to an existing node or arbitrary point, use the following procedure.

1. Select one or more members in the view window.
2. On the Specification ribbon tab, select the Beam > Beam Reference Point tool in the Specifications group.

   The Referene Point dialog opens.

3. Select the type of point to use as a reference for orientation:
   - **To orient by...**
   - **Select the following...**
   - an arbitrary point in space
     select the Point option and define the coordinates with respect to the model origin
To orient by...

- **a vector from the member start node**
  - select the **Vector** option and then define the second point of the vector coordinates with respect to the member start node
- **a node in the model**
  - select the **Node** option and then select the node number from the drop-down list

4. Select the **To Selection** option.

   **Tip:** The **To View** option assigns the specification to all members in the view window.

5. Click **OK**

**Related Links**
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2503)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2301)
- *Reference Point dialog* (on page 2979)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2503)

**M. Member Specifications**

**M. To add a member specification**

To add a specification for beam members in your model, use the following procedure.

   **Tip:** As with most assignment options, you can select a set of objects prior to starting this procedure and make the assignments to this selection set by using the dialog's **Assign** button.

Member specifications are used to assign beam members as cables, truss, tension- or compression-only. Similarly, additional specifications for member end offsets, fireproofing, and inactive members can be assigned.

1. Select the **Specifications** page in the Analytical Modeling page control bar.
   - The **Specifications - Whole Structure** dialog opens.
2. In the **Specifications - Whole Structure** dialog, click **Beam**.
   - The **Member Specification** dialog opens.
3. Select the dialog tab corresponding to the member specification you want to add:
   - Release
   - Offset
   - Property Reduction Factor
   - Cable
   - Truss
   - Compression
   - Tension
   - Inactive
   - Fire Proofing
   - Imperfection
4. Either:

To...

<table>
<thead>
<tr>
<th>Do the following...</th>
</tr>
</thead>
<tbody>
<tr>
<td>add the specification to the model for later assignment</td>
</tr>
<tr>
<td>add the specification to the model and assign to the current member selection</td>
</tr>
</tbody>
</table>

The dialog closes.

The member specification is added to the **Specifications - Whole Structure** dialog and the **Specification Values** table.

You can repeat this procedure as many times as necessary to add additional member specifications to your model.

If you did not assign the specification to a selection set, you must select the specification in the **Specifications - Whole Structure** dialog and assign it using one of the assignment methods.

**Related Links**
- **Specifications - Whole Structure dialog** (on page 2959)
- **Member Specification dialog** (on page 2962)

---

**M. To assign axial action members**

To assign a member as compression-only, tension-only, or a truss (axial-only) member, use the following procedure.

STAAD.Pro allows you to specify the axial actions of members.

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam** tool in the **Specification** group.

A list of member specification types opens.

3. Select the specification type you want to assign:
   - **Compression Only** – members are capable of carrying compression forces only
   - **Tension Only** – members are capable of carrying tensile forces only
   - **Truss** – members are capable of carrying axial forces only

   **Note:** These member specifications have no additional parameters.

The **Member Specification** dialog opens to the corresponding tab.

4. Either:
To...
add the specification to the model and assign to the current member selection   click Assign.
add the specification to the model for later assignment   click Add.

The dialog closes.

The member specification is added to the Specifications - Whole Structure dialog and the Specification Values table.

Related Links
• *G.8.1 Truss and Tension- or Compression-Only Members* (on page 2330)
• *TR.23.1 Member Truss Specification* (on page 2492)
• *TR.23.3 Member Tension/Compression Specification* (on page 2495)
• *Member Specification dialog* (on page 2962)

M. To assign member end release

To assign a member end release or partial member end release, use the following procedure.

Release specifications are created for each of a member separately. To release both ends, you must create two specifications.

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the Specifications ribbon tab, select the Beam > Release tool in the Specifications group.

   The Member Specification dialog opens to the Release tab.
3. Select which location on the member to which the release applies: Start or End.
4. Select the Release Type:

   Partial Moment Release
   
   or
   
   Release

5. For Partial Moment Release, select the directions which are partially released and the release ratio:
   a. Check the boxes corresponding to the directions of release.
      Check MP to assign the same partial moment release in all three local directions. Check a combination of MPX, MPY, and MPZ to apply different partial moment releases in each direction.
   b. For each direction checked, type a release ratio.
      This is a ratio between 0 and 1, where 0 indicated full moment restraint and 1 indicates full release.

6. For full Release, check the release directions which apply.
   These are FX, FY, FZ, MX, MY, and MZ. Any directions not checked will be fully restrained in that degree of freedom.

7. (Optional) To specify a spring in any degree of freedom, check the corresponding Release spring direction and type a spring constant in the adjacent field (in the indicated units).
   Springs can be applied to partial moment release or release types.
8. Either:
To…

<table>
<thead>
<tr>
<th>To…</th>
<th>Do the following…</th>
</tr>
</thead>
<tbody>
<tr>
<td>add the specification to the model and assign to the current member selection</td>
<td>click Assign.</td>
</tr>
<tr>
<td>add the specification to the model for later assignment</td>
<td>click Add.</td>
</tr>
</tbody>
</table>

The dialog closes.

Related Links

- *G.7 Member and Element Release* (on page 2329)
- *TR.22.1 Member Release Specification* (on page 2488)
- *Member Specification dialog* (on page 2962)

**M. To assign member end offsets**

To assign a rigid end offset to one end of a member, use the following procedure.

Member end offsets can be used to model any situation where the end of the actual member does not coincide with the analytical node used for the member end incidence. For example,

- working points that do not align (e.g., a gusset plate connection)
- rigid end zones (e.g., a beam that connects to the flange of a stiff column)
- modeling top of steel (e.g., the center line of a beam that supports a slab)

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam > Offset** tool in the **Specification** group.

The **Member Specification** dialog opens to the **Offset** tab.

3. Select which **Location** on the member to which the offset applies: **Start** or **End**.
4. Select which **Direction** reference to use for the offset values: **Global** or **Local**.
5. Type the **Offsets** to use along each of the **X**, **Y**, and **Z** axes.
6. Either:

To…

<table>
<thead>
<tr>
<th>To…</th>
<th>Do the following…</th>
</tr>
</thead>
<tbody>
<tr>
<td>add the specification to the model and assign to the current member selection</td>
<td>click Assign.</td>
</tr>
<tr>
<td>add the specification to the model for later assignment</td>
<td>click Add.</td>
</tr>
</tbody>
</table>

The dialog closes.

Related Links

- *G.11 Member Offsets* (on page 2334)
- *TR.25 Member Offset Specification* (on page 2499)
- *Member Specification dialog* (on page 2962)

**M. To assign member imperfection for members**

To assign member drift or camber for use with an imperfection analysis, use the following procedure.
1. (Optional) Select the members which will all have the same member specification assigned.

2. On the **Specification** ribbon tab, select the **Beam > Imperfection** tool in the **Specification** group.

   ![Image]

   The **Member Specification** dialog opens to the **Imperfection** tab.

3. Do either of the following:
   
   select **Camber** to define the maximum offset along a beam from a line connecting the end points (typically for beams)
   
   or
   
   select **Drift** to define the offset at the end of a member (typically for columns)

4. Select the **Local Direction** of the imperfection and then type the **Value**, which is taken as a ratio of the member length to offset (i.e., L/d).

5. (Optional) For Camber, type a **Respect** value which is used to determine when to skip camber imperfection calculations.

   This ratio results in the camber imperfection calculations being skipped when:
   
   - the compressive load is small,
   - the member stiffness ($EI$) is large, or
   - the member is short

6. Either:

   **To...** 
   
   **Do the following...**
   
   - add the specification to the model and assign to the current member selection 
     
     click **Assign**.
   
   - add the specification to the model for later assignment
     
     click **Add**.

   The dialog closes.

You must assign the member imperfection specification to one or more members. An imperfection analysis is required in order to perform these calculations.

**Related Links**

- *TR.26.6 Member Imperfection Information* (on page 2511)
- *Member Specification dialog* (on page 2962)
- *Perform Imperfection Analysis tab* (on page 3073)
- *TR.26.6 Member Imperfection Information* (on page 2511)

**M. To assign nonlinear cable members**

1. (Optional) Select the members which will all have the same member specification assigned.

2. On the **Specification** ribbon tab, select the **Beam > Cable** tool in the **Specification** group.

   ![Image]

   The **Member Specification** dialog opens to the **Cable** tab.

3. (Optional) Do either of the following:
select the **Initial TENSION** option and then type a value for the tension force in the cable

or

select the **Unstressed TENSION** option and then type a value for the initial length of the cable

If no value is given for either option, then a minimal cable tension is assumed.

4. Type a ratio value for one or more of the **Factor in global X : Fwx**, **Factor in global Y : Fwy**, or **Factor in global Z : Fwz**.

**Note:** These loads are used for Advanced Cable Analysis only.

The values are a multiplier of the self weight of the cable, applied in the selected global direction.

5. Either:

   **To...**                 **Do the following...**
   
   - add the specification to the model and assign to the current member selection
     
     click **Assign**.
   
   - add the specification to the model for later assignment
     
     click **Add**.

The dialog closes.

**Related Links**

- **TR.23.2 Member Cable Specification** (on page 2493)
- **Member Specification dialog** (on page 2962)

---

**M. To assign cracked section properties to a member**

To assign reduced section property factors to a member, use the following procedure.

Global reduction factors will be applied to any member, regardless of material. Code-specific reduction factors are only applied to concrete members (non-concrete members included in the assignment are ignored).

**Note:** Automated stiffness reduction analysis is not supported by the basic solver. STAAD.Pro Advanced is required for this feature.

**Note:** For reducing the stiffness of a steel member for direct analysis per AISC 360, it is recommended use the REDUCEDEI parameter in the steel design parameters instead.

**Notes:** Reduction factors are considered for analysis only but not for design.

Results using the reduced section properties are not available when using the member query feature.

1. (Optional) Select the members which will all have the same member specification assigned.

2. On the **Specification** ribbon tab, select the **Beam > Cracked Property** tool in the **Specification** group.

   ![Member Specification dialog](image)

   The **Member Specification** dialog opens to the **Property Reduction Factors** tab.

3. Select either:

   **Global** – specify the **Reduction Factors** to be used below, which will apply to any member regardless of material.
or

**Code Specific** – select the building code to use reduction factors specific to load cases per that code which are only applied to concrete members:

**IS1893 2016**

4. Type the reduction factor values to assign to the current member selection:

- Reduction Factor for Cross sectional Area (RAX)

  **Note:** For IS1893 2016 reduction factors, the RAX input field is inactive as this property reduction is not mandated by that code for concrete members.

- Reduction Factor for Torsion Constant (RIX)
- Reduction Factor for Moment of Inertia, major axis (RIY)
- Reduction Factor for Moment of Inertia, minor axis (RIZ)

  **Note:** Reduction factor values should be between 0 and 1 (inclusive).

5. Either:

   - To... 
   - Do the following...

     - **add the specification to the model and assign to the current member selection**
     - **click Assign.**

     - **add the specification to the model for later assignment**
     - **click Add.**

The dialog closes.

**Notes:** Reduction factors are considered for analysis only but not for design.

Results using the reduced section properties are not available when using the member query feature.

**Related Links**

- **Member Specification dialog** (on page 2962)
- **TR.20.10 Member Property Reduction Factors** (on page 2485)
- **TR.20.10 Member Property Reduction Factors** (on page 2485)
- **Member Specification dialog** (on page 2962)

**M. To assign member fire proofing**

To assign fire proofing to a member, use the following procedure.

**Tip:** Fire proofing thickness is specified in the current units of distance. You may want to change the units to a convenient value before beginning.

The fire proofing specification allows the program to automatically calculate the weight of the fire proofing material applied to members.

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam > Fire Proofing** tool in the **Specification** group.
The **Member Specification** dialog opens to the **Fire Proofing** tab.

3. Select the **Fire Proofing Type**:
   - **BFP (Block Fireproofing):**
     The fire-protection material forms a rectangular block around the steel section. The thickness specified is the minimum thickness which defines the outer block dimensions.
     ![](image1)
   - **CFP (Contour Fireproofing):**
     The fire-protection material forms a coating around the steel section. The thickness specified is a constant thickness around the section profile.
     ![](image2)

4. Type the **Thickness** of the fire proofing in the units indicated.
5. Type the **Density** of the fire proofing material.
6. Either:
   - **To...**
   - **Do the following...**
     - add the specification to the model and assign to the current member selection
       - click **Assign**.
     - add the specification to the model for later assignment
       - click **Add**.

   The dialog closes.

**Related Links**
- TR.20.9 Applying Fireproofing on members (on page 2482)
- Member Specification dialog (on page 2962)

**M. To assign specifications to physical members**

To assign member end or axial specifications to physical members, use the following procedure.

Physical members can have member end releases, member end offsets, and truss specifications. Other analytical member specifications do not apply to physical members.
Note: For a physical member whose physical member specification is already assigned, the individual analytical members in that Physical member will adopt the same member specification that of the physical member. However, in case, where the analytical member specification is assigned to any member in that physical member, then this analytical member specification will supersede physical member specification.

2. On the Geometry ribbon tab, select the Toggle Physical Member Mode tool in the Physical Members group.
   
   The tool is highlighted to indicate the mode is active.
   
   Note: The dialog only shows three tabs in the physical modeling mode.
4. Select the specification type you want to assign:
   
   Release – one or more degrees of freedom are free or partially restrained at one end of the member
   
   or
   
   Offset – a rigid offset is present between one end of the member and the joint
   
   or
   
   Truss – members are capable of carrying axial forces only
5. Add Release or Offset attributes as you would for analytical members.
   
   a. Select the Location on the member: Start or End.
   
   b. For a Release, select if either partial or full release and the degrees of freedom of the release.
   
   c. For an Offset, select the directions of the offset.
6. Click Add.

Related Links

- TR.23.1 Member Truss Specification (on page 2492)
- TR.25 Member Offset Specification (on page 2499)
- TR.22.1 Member Release Specification (on page 2488)

M. Plate Specifications

M. To align a plate to a reference point

To orient the local axis of a plate element towards or away from an arbitrary point, use the following procedure.

The local z axis of the plates will remain perpendicular to the plane of the elements. This tool will simply orient the local axis such that this z axis points generally toward or away from the specified reference point. Refer to Plate Element Local Coordinate System (on page 2308) for additional details.
1. Select one or more plates in the view window.
2. On the Specification ribbon tab, select the Plate > Plate Reference Point tool in the Specifications group.

   The Plate Reference Point dialog opens.

3. Specify the X, Y, and Z coordinates of an arbitrary reference point.
4. Select if the Local Z Axis of the elements should point towards or away from the reference point.

   ![Diagram of plate reference point](image)

   *Figure 115: The local z axis of the plate is perpendicular to the plane of the element*

5. Select the To Selection option.

   **Note:** The To View option assigns the specification to all plates in the view window.

6. Click OK

**Related Links**

- G.5.1 Plate and Shell Elements (on page 2308)
- Plate Reference Point dialog (on page 2981)

**M. To specify plate thickness**

To specify the thickness of a plate element, use the following procedure.

1. Select the plates which will have a similar thickness.
2. On the Specification ribbon tab, select the Plate Thickness tool in the Plate Profiles group.

   The Properties - Whole Structure and the Plate Element/ Surface Property dialogs open.
3. Select the Plate Element Thickness tab.
4. Type the thickness values for the plate:
   
   **For a...**  **Do this...**
   
   uniform plate thickness  type the thickness in the Node 1 field. The other node fields will use this value by default
   
   linearly varying thickness  type the thickness of the plate at each corner node in the corresponding field

5. (Optional) Check the **Material** option and select the material definition from the drop-down list.
6. Either:
   
   click **Assign** to assign the thickness (and optional material) to the selection set
   
   or
   
   click **Add** to add the thickness property to the model for assignment later

**Related Links**

- *G.5.1 Plate and Shell Elements* (on page 2308)
- *TR.21.1 Element Property Specification* (on page 2487)
- *Plate ElementProperty dialog* (on page 2981)

**M. To assign plate corner release**

To release degrees of freedom of a plate element corner node, use the following procedure.
Releases are created for each node (by reference order) separately. To release multiple corners, you must create multiple specifications.

1. (Optional) Select the plates which will have the same plate release specifications assigned.
2. On the Specification ribbon tab, select the Plate > Release tool in the Specifications group.

   The Plate Specs dialog opens to the Release tab.
3. Select which Node to which the release applies.
   These are based on the local coordinate system for the element.
   
   **Tip:** To quickly identify the order of element corners, use the Plate Cursor tool and double click a plate. The Plate query dialog displays the nodes in order on the Geometry tab.
4. Check the Release directions.
   These are FX, FY, FZ, MX, MY, and MZ. Any directions not checked will be fully restrained in that degree of freedom.
5. Either:
   
   **To...**
   
   **Do the following...**
   
   add the specification to the model and assign to the current plate selection  
   click Assign.
   
   add the specification to the model for later assignment  
   click Add.

   The dialog closes.

**Related Links**

- [G.7 Member and Element Release](on page 2329)
- [Plate Specs dialog](on page 2967)
- [TR.22.2 Element Release Specification](on page 2490)

**M. To assign plates as plane stress**

To assign plate elements as plane stress elements, use the following procedure.

This specification results in plates that only resist stress within the plane of the plate, but resist no out-of-plane bending. These plate elements are analogous to truss members.

   **Tip:** You can also assign this plate specification from the Plate Specs dialog, which opens when you click Plate on the Specifications - Whole Structure dialog.

1. Select the plates which will be plane stress elements.
2. On the Specification ribbon tab, select the Plate > Plane Stress tool in the Specifications group.

   The Plate Specs dialog opens to the Plane Stress tab.
Note: There are no parameters to provide for this plate specification.

3. Either:
   
   To... Do the following...
   
   add the specification to the model and assign to the current plate selection click Assign.
   
   add the specification to the model for later assignment click Add.

   The dialog closes.

Related Links
- G.5.1 Plate and Shell Elements (on page 2308)
- Plate Specs dialog (on page 2967)
- TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2498)

M. To assign inplane rotation behavior to plates

To specify either no inplane rotational stiffness or completely rigid inplane rotational stiffness, use the following procedure.

Typically, the plate element used in STAAD.Pro has a very “soft” in-plane rotational, Mz, stiffness. There may be circumstances where this rotation should be ignored entirely (that is, with zero in-plane rotational stiffness). Alternatively, there may be circumstances where a rigid body rotation is required in plate elements.

1. Select the plates which will have the same inplane rigidity.
2. On the Specification ribbon tab, in the Specifications group, select either:

   To... Select this tool...
   
   ignore the in-plane rotation actions of the plates Plate > Ignore Inplane Rotation
   
   treat the plates as rigid bodies for in-plane rotation Plate > Rigid Inplane Rotation

   The Plate Specs dialog opens to the corresponding tab.

   Note: There are no parameters to provide for this plate specification.

3. Either:
   
   To... Do the following...
   
   add the specification to the model and assign to the current plate selection click Assign.
   
   add the specification to the model for later assignment click Add.

   The dialog closes.

Related Links
- G.5.1 Plate and Shell Elements (on page 2308)
- Plate Specs dialog (on page 2967)
- TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2498)
M. To ignore plate stiffness

To ignore the stiffness of a selection of plates, use the following procedure.

A plate element whose stiffness is ignored by the program acts will distribute any element loads but the stiffness of the plate does not contribute to the stiffness of the structure. In this way, plate elements can be used to model non-structural components, such as glass cladding.

Tip: You can also assign this plate specification from the **Plate Specs** dialog, which opens when you click **Plate** on the **Specifications - Whole Structure** dialog.

1. Select the plates which will have their stiffness ignored by the program.
2. On the **Specification** ribbon tab, select the **Plate > Ignore Stiffness** tool in the **Specifications** group.

The **Plate Specs** dialog opens to the **Ignore Stiffness** tab.

**Note:** There are no parameters to provide for this plate specification.

3. Either:
   - To...                        Do the following...
     - add the specification to the model and assign to the current plate selection
       - click **Assign**.
     - add the specification to the model for later assignment
       - click **Add**.

The dialog closes.

**Related Links**
- *G.5.1 Plate and Shell Elements* (on page 2308)
- *Plate Specs dialog* (on page 2967)
- *TR.22.3 Element Ignore Stiffness* (on page 2491)

M. Node Specifications

M. To assign a rigid link between nodes

To create a rigid link between a master node any number of subordinate nodes, use the following procedure.

A master/slave specification establishes a primary node and a set of one or more subordinate nodes (i.e., “slave nodes”). Any displacements or rotations along the specified direction at the primary node will be directly translated to the subordinate nodes.

1. Select the nodes that will be the subordinate nodes.
2. On the **Specifications** ribbon tab, select the **Nodes Add Master/Slave Specification** tool in the **Specifications** group.
The Node Specification dialog opens to the Master/Save tab.

3. Select the Master Node number from the drop-down list of nodes.

4. Select a rigid link specification in the Slaved Directions:
   - Rigid link type Select...
   - in all directions the Rigid option
   - within a plane either the XY, YZ, or XZ option, corresponding to the plane
   - in specified degrees of freedom the specific degrees of freedom which are linked between the primary and subordinate nodes: FX, FY, FZ, MX, MY, and MZ

5. Either:
   - To add the specification to the model and assign to the current node selection click Assign.
   - add the specification to the model for later assignment click Add.

The dialog closes.

Related Links
- G.14 Rigid Diaphragms (on page 2336)
- TR.28.1 Master/Slave Specification (on page 2525)
- Node Specification dialog (on page 2960)

M. To assign nodes to a floor diaphragm

To define nodes that make up a floor diaphragm, use the following procedure.

Tip: Diaphragm heights and optional ranges are given in the current units of length. You may want to change the units of length to a convenient value before defining the diaphragm.

In order to generate the center of gravity for a diaphragm, you must create a mass load case which includes the weights to be considered for the diaphragm.

1. (Optional) Select a set of nodes which will be used to define the diaphragm.
2. Select the Specifications page in the Analytical Modeling page control bar.
   The Specifications - Whole Structure dialog opens.
   The Node Specification dialog opens.
4. Select the Floor Diaphragm tab.
5. Specify the floor level of the diaphragm by either:
   - selecting Height and then typing the height of the floor (Y coordinate value)
   - or
   - selecting YRange and then typing the Minimum and Maximum values of the diaphragm (Y coordinates)
6. (Optional) Select the Define Floor Range option to specify a boundary in the XZ plane of the diaphragm to define its edges:
select the **Select Nodes** option to either assign the diaphragm to the current node selection or to assign it later

or

select the **Floor Range** option to type maximum and minimum values in the global X and Z directions to bound the diaphragm.

Nodes lying outside of this selection or range will not be considered to be part of the diaphragm even if they are at the diaphragm height.

7. (Optional) You can specify a master node to use as the center of gravity by selecting the **Select Master Node** option and then selecting the node number to use as the center of gravity.

Otherwise, the program will calculate the center of gravity location for the diaphragm and add an analytical node at this location.

**Tip:** To report calculated CGs, add the Cg (Center of Gravity) option in the Post-Analysis Print Commands.

8. Either:

To... Do the following...

add the specification to the model and assign to the diaphragm to the current nodes click **Assign**.

add the specification to the model for later assignment or to use all nodes at the specified Height click **Add**.

The dialog closes.

**Note:** Soft story checks can be made for structures with rigid floor diaphragms per the IS 1893 2002, IS 1893 2016, and ASCE 7 specifications.

**Related Links**

- *TR.28.2 Floor Diaphragm* (on page 2526)
- *Node Specification dialog* (on page 2960)

**M. Supports**

This section describes how to create boundary conditions for your model.

Select the **Supports** page in the in the **Analytical Modeling** workflow to open the **Supports - Whole Structure** dialog.

In STAAD.Pro, you will create support types that can be assigned to multiple nodes.

**M. To assign a fixed or pinned support**

To specify a node as either a fixed or pinned support, use the following procedure.

A fixed support is restrained against movement (translation and rotation) in all degrees of freedom. A pinned support is restrained against translation only, but is otherwise free to rotate. Refer to *Description of Pinned and Fixed* (on page 2514) for additional details.

1. Select the nodes that will have the same support condition.
2. On the Specifications ribbon tab, select one of the following tools in the Supports group:

![Fixed Support](image)

or

![Pinned Support](image)

3. The Create Support dialog opens to the corresponding tab.

4. Either:

   To...  
   
   add the support type to the model and assign to the current node selection  
   click Assign.

   add the support to the model for later assignment  
   click Add.

The dialog closes.

Related Links

- G.13 Supports (on page 2335)
- Create Support dialog (on page 2983)
- Supports - Whole Structure dialog (on page 2982)
- TR.27.1 Global Support Specification (on page 2514)

M. To assign an enforced support

To assign a support which can have a support displacement imposed as a load, use the following procedure.

In STAAD.Pro, a special support type is used for supports that will have an imposed displacement as part of a load case. These are called “enforced” supports.

**Note:** If no support displacements are defined for an enforced support, then this support behaves as a fixed support (i.e., no translation or rotation). Similarly, an enforced but support behaves like a corresponding fixed but support if no support displacements are defined.

1. Select the nodes that will have the same support condition.
2. On the Specifications ribbon tab, select the Other Supports tool in the Supports group.
3. Select one of the drop-down support types as follows:

<table>
<thead>
<tr>
<th>Support condition...</th>
<th>Select the following other support...</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixed in all degrees of freedom</td>
<td>Enforced</td>
</tr>
<tr>
<td>released in one or more degrees of freedom</td>
<td>Custom Enforced</td>
</tr>
</tbody>
</table>

To impose a displacement on a joint with releases (e.g., a “pinned” support), you must use a custom enforced (i.e., ENFORCED BUT) support type and define the released degrees of freedom.

The Create Supports dialog opens to the corresponding tab.

4. (Optional) For an enforced but support type, check the Releases for this support type.
   For example, to create a “pinned” type support, you would check the MX, MY, and MZ release options.

5. Either:
To...

add the support type to the model and assign to the current node selection  
add the support to the model for later assignment

Do the following...

click Assign.

click Add.

The dialog closes.

You must create a load case with support displacements defined at nodes with enforced or enforced but support types assigned in order for this support type to have an effect on your model beyond behaving as a corresponding fixed or fixed but support.

Related Links

• G.13 Supports (on page 2335)
• EX. US-24 Analysis of a Concrete Block Using Solid Elements (on page 4552)
• EX. UK-24 Analysis of a Concrete Block Using Solid Elements (on page 4833)
• TR.27.1 Global Support Specification (on page 2514)
• Create Support dialog (on page 2983)

M. To assign custom release supports

To assign a set of custom releases to a supported node, use the following procedure.

1. Select the nodes that will have the same support condition.
2. On the Specifications ribbon tab, select the Custom tool in the Supports group.

The Create Supports dialog opens to the Fixed But tab.

3. In the Releases group, check the degrees of freedom to be released for this support type.

   Note: If a spring constant is specified in a direction, then the corresponding degree of freedom cannot be released. A given support type can have a mix of releases and springs, but only in different degrees of freedom.

4. Either:

   To...

   add the support type to the model and assign to the current node selection  
   add the support to the model for later assignment

   Do the following...

   click Assign.

   click Add.

The dialog closes.

M. To assign a spring support

To assign a linear spring in one or more degrees of freedom to a support, use the following procedure.

You may want to change the system of input units to convenient values for your spring constants before creating a spring support type.

In STAAD.Pro, linear spring constants are specified as part of “fixed but” support types.
1. Select the nodes that will have the same support condition.
2. On the Specifications ribbon tab, select the Custom tool in the Supports group.

The Create Supports dialog opens to the Fixed But tab.

3. In the Define Springs group, type the linear spring constant to use for each translational and rotational spring direction.

   - **KFX** - translation in the global x direction
   - **KFY** - translation in the global y direction
   - **KFZ** - translation in the global z direction
   - **KMX** - rotation about the global x axis
   - **KMY** - rotation about the global y axis
   - **KMZ** - rotation about the global z axis

   **Note:** If a spring constant is specified in a direction, then the corresponding degree of freedom cannot be released. A given support type can have a mix of releases and springs, but only in different degrees of freedom.

   For example, a fixed but support could have MX, MY, and MZ released as in a pinned support and also have spring constants defined in FX, FY, and FZ.

4. Either:
   - **To...**
     - add the support type to the model and assign to the current node selection
     - add the support to the model for later assignment
   - **Do the following...**
     - click Assign.
     - click Add.

   The dialog closes.

You can now assign tension-only, compression-only, or multi-linear springs to these same supported nodes if necessary.

**Related Links**
- G.13 Supports (on page 2335)
- EX. US-3 Soil Springs for Portal Frame (on page 4383)
- EX. UK-3 Soil Springs for Portal Frame (on page 4663)
- TR.27.1 Global Support Specification (on page 2514)
- Create Support dialog (on page 2983)

**M. To assign multilinear springs to spring supports**

To model varying resistance to external loads at a spring support, use the following procedure.

You must assign spring supports to nodes before assigning multilinear spring definitions. You may not use multilinear springs in a model that contains tension-only or compression-only springs.

You may want to change the system of input units to convenient values for your spring constants before creating a spring support type.
**Note:** Multilinear springs can only be used with a select set of analysis types. Each load case should be separated by a Change command as well. See TR.27.4 Multilinear Spring Support Specification (on page 2520) for details.

1. Select one or more nodes assigned with spring support types.
2. On the **Specifications** ribbon tab, select the **Other Supports > Multi-Linear Spring** tool in the **Supports** group.
   - The **Create Support** dialog opens to the **Multilinear Spring** tab.
3. In the table, type the Spring Stiffness constant which corresponds to a Displacement.
   - These should be in order starting with the minimum displacement (e.g., this may also be the largest negative displacement) to the maximum positive displacement. All spring constants should be zero or positive values.
4. Click **Assign**.

### To assign support springs as a one-way

To specify support supported nodes as capable of carrying only tension or compression, use the following procedure.

You must assign spring supports to nodes before assigning tension-only or compression-only spring definitions. You may not use tension-only or compression-only springs in a model that contains multilinear springs.

STAAD.Pro assumes that negative displacement is compression and positive displacement is tension.

1. Select one or more nodes assigned with spring support types.
2. On the **Specifications** ribbon tab, select the **One-way Spring** tool in the **Supports** group.

   - The **Create Support** dialog opens to the **Tension/Compression Only Springs** tab.
3. Select the load direction the support is capable of resisting:
   - Tension Only
   - or
   - Compression Only
4. Select the Spring Direction options for this support type:
   - KFX
   - KFY
   - KFZ

   **Note:** One-way action is for translational springs only.
5. Click **Assign**.

**Related Links**
- TR.27.5 Spring Tension/Compression Specification (on page 2522)
- Create Support dialog (on page 2983)
M. To assign an inclined support

To assign a support that acts inclined with respect to the global axis, use the following procedure.

You may want to change the length unit to a more convenient value if you are going to define an incline direction by coordinate or vector.

See Inclined Support Axis System (on page 2516) for details on how the reaction directions are determined from the reference coordinate.

1. Select the nodes that will have the same support condition.
2. On the Specification ribbon tab, select the Other Supports > Inclined tool in the Supports group. The Create Support dialog opens to the Inclined tab.
3. Select the method for specifying the Incline Reference Point:

   To use this reference…  Do…
   
an arbitrary point with respect to the supported node  select the Coordinate option and then type the relative global X, Y, and Z distances to the reference point
   
an arbitrary point with respect to the model origin  select the Ref option and then type the global X, Y, and Z coordinates
   
an existing node in the model  select the RefJt option and then select the node number from the drop-down list

   **Note:** If you select the Ref or RefJt options, then the orientation of the inclined support axis system may vary from node to node.

4. Select the Support Type to use.

   - **Pinned** - restrained against translation but free to rotate
   - **Fixed** - restrained in all degrees of freedom
   - **Fixed But** - released in specified degrees of freedom and can have spring constants assigned in restrained degrees of freedom
   - **Enforced** - restrained in all degrees of freedom and can have imposed support displacements assigned as loads
   - **Enforced But** - released in specified degrees of freedom and can have imposed support displacements assigned as loads

5. (Optional) For Fixed But support type, type linear spring constants to use as necessary.
6. For Fixed But or Enforced But support types, select the released degrees of freedom.
7. Either:

   To…  Do the following…
   
   add the support type to the model and assign to the current node selection  click Assign.
   
   add the support to the model for later assignment  click Add.

   The dialog closes.

**Related Links**

- **G.13 Supports** (on page 2335)
- **EX. US-19 Inclined Supports** (on page 4515)
To assign a foundation support

To automatically model a support using foundation parameters, use the following procedure.

Isolated footing dimensions and subgrade modulus values are specified in the current input units. You may want to change these to a convenient value before specifying a foundation support.

STAAD.Pro can take foundation parameters for footings or mat foundations and automatically model spring supports.

1. Select the nodes that will have the same support condition.
2. On the Specification ribbon tab, select the Foundation tool in the Supports group.

The Create Support dialog opens to the Foundation tab.

3. Select the Foundation type to model:
   - **Footing** - a rectangular, isolated footing (i.e., spread footing) area is used to model spring stiffness using the subgrade modulus
     or
   - **Elastic Mat** - the program calculates the influence area of the load and then calculates a spring stiffness based on this area using the subgrade modulus
     or
   - **Plate Mat** - the program calculates the influence area of the load within a specified plate and then calculates a spring stiffness based on this area using the subgrade modulus

For additional details on the foundation types and their applicability, refer to **TR.27.3 Automatic Spring Support Generator for Foundations** (on page 2517).

4. Select the Direction of the resistance of the spring supports.

   The directions X, Y, or Z generate a spring in that global direction. The other two global directions are fixed against translation and moment about the selected direction is also fixed. The directions X Only, Y Only, or Z Only generate a spring in that global direction only.

5. Type the Subgrade Modulus of the soil.

6. (Optional) For Elastic Mat or Plate Mat foundation types, check the option Print influence area of each joint to include this information in the output.

7. (Optional) For Elastic Mat or Plate Mat foundation types, select a nonlinear spring option if necessary:

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td>linear springs are generated</td>
</tr>
<tr>
<td>Compression Only</td>
<td>generated springs are compression only (made inactive if in tension)</td>
</tr>
<tr>
<td>Multi-Linear</td>
<td>generated springs can have a multilinear displacement-spring constant curve associated with the mat foundation</td>
</tr>
</tbody>
</table>
Tip: Multilinear springs must be specified using the STAAD Editor. The dialog box cannot accept displacement-spring constant curve data.

8. Either:

<table>
<thead>
<tr>
<th>To...</th>
<th>Do the following...</th>
</tr>
</thead>
<tbody>
<tr>
<td>add the support type to the model and assign to the current node selection</td>
<td>click Assign.</td>
</tr>
<tr>
<td>add the support to the model for later assignment</td>
<td>click Add.</td>
</tr>
</tbody>
</table>

The dialog closes.

Related Links

- *G.13 Supports* (on page 2335)
- *EX. US-23 Spring Support Generation for a Slab on Grade* (on page 4541)
- *EX. UK-23 Spring Support Generation for a Slab on Grade* (on page 4822)
- *TR.27.3 Automatic Spring Support Generator for Foundations* (on page 2517)
- *Create Support dialog* (on page 2983)

M. Loading Your Model

Load items in STAAD.Pro follow the general assigning workflow: you will define the load items within a primary load case. Then you will assign these load items to structural objects.

M. Available Structural Load Specifications in STAAD.Pro

The program contains the following load specifications:

Response Spectrum

**Table 43: Codes available in STAAD.Pro with Response Spectrum loads**

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Canada</td>
<td><strong>NRC 2005</strong></td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td></td>
<td>(on page 2694)</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>NRC 2010</strong></td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td></td>
<td>(on page 2702)</td>
<td></td>
</tr>
<tr>
<td>Country</td>
<td>Code</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>-------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Russia</td>
<td>SNiP II-7-81 (on page 2759)</td>
<td>Строительство в сейсмических районах (Construction in Seismic Regions)</td>
</tr>
<tr>
<td></td>
<td>SP 14.13330.2011 (on page 2764)</td>
<td>Строительство в сейсмических районах (Construction in Seismic Regions)</td>
</tr>
</tbody>
</table>

**Seismic**

**Table 44: Codes available in STAAD.Pro with Seismic loads**

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Algeria</td>
<td>RPA 99</td>
<td>Règles Parasismiques Algériennes</td>
</tr>
<tr>
<td>Canada</td>
<td>NRC 1995</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td></td>
<td>NRC 2005</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td></td>
<td>NRC 2010</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td>Country</td>
<td>Code</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>------</td>
<td>-------</td>
</tr>
<tr>
<td>Colombia</td>
<td>Colombian (on page 2571)</td>
<td>Reglamento Colombiano de Construcción Sismo Resistente (NSR-98), Normas Colombianas de Diseño y Construcción, 1998, Asociación Colombiana de Ingeniería Sísmica</td>
</tr>
<tr>
<td></td>
<td>Colombian 2010 (on page 2573)</td>
<td>NSR-10 Reglamento Colombiano Sismo Resistente</td>
</tr>
<tr>
<td>India</td>
<td>IS:1893 1984 (on page 2576)</td>
<td>Criteria for Earthquake Resistant Design of Structures</td>
</tr>
<tr>
<td>Japan</td>
<td>AIJ 2006 (on page 2605)</td>
<td>Building Codes Enforcement Ordinance 2006</td>
</tr>
<tr>
<td></td>
<td>NTC (on page 2610)</td>
<td>Reglamento de Construcciones del Distrito Federal de México (Mexico Federal District)</td>
</tr>
<tr>
<td>Turkey</td>
<td>Turkish (on page 2613)</td>
<td>“Specification for Structures to be Built in Disaster Areas Part – III – Earthquake Disaster Prevention” Amended on 2.7.1998, Official Gazette No. 23390</td>
</tr>
</tbody>
</table>
Wind

Table 45: Codes available in STAAD.Pro with Wind loads

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Russia</td>
<td>SNiP 2.01.07-85</td>
<td>Loads and Actions (1985)</td>
</tr>
<tr>
<td></td>
<td>ASCE 7-2002</td>
<td>Minimum Design Loads for Buildings and Other Structures</td>
</tr>
<tr>
<td></td>
<td>ASCE 7-2010</td>
<td>Minimum Design Loads for Buildings and Other Structures</td>
</tr>
</tbody>
</table>

M. To create a new primary load case

This procedure applies to the Loading page in the Analytical Modeling workflow.

1. Either:
select the **Primary Load Cases** tool in the **Loading** group on the **Loading** ribbon tab

or

select the **Load Cases Details** section on the **Load & Definitions** dialog and then click **Add**

The Add New Load Cases dialog opens.

2. Select the **Primary** tab.
3. (Optional) Type a Number for the load case.

   The program will increment the load case number based on existing load case numbers.
4. (Optional) Select the **Loading Type**.

   This selection is used when automatically generating load combinations. If you select a live load type, you may also indicated if the load is **Reducible per UBC/IBC**.
5. Type a **Title** used to easily identify this load case.
6. Click **Add**.

   The empty load case is added to the **Load Cases Details** list.

You must now add load items to this load case and assign them to objects in the structure.

You can edit the load case title and loading type by selecting the entry in the **Load Cases Details** list and then clicking **Edit** in the **Load & Definitions** dialog.

You can remove the load case (along with any associated load items) by selecting the entry in the **Load Cases Details** list and then clicking **Delete** in the **Load & Definitions** dialog.

**Related Links**
- **Create Primary Load Case dialog** (on page 3002)
- **TR.32 Loading Specifications** (on page 2650)

**M. Load Items**

Individual load items are added to primary load cases.

**M. To add selfweight load**

To add the calculated weight of the structure objects as a load item, use the following procedure.

You must create a primary load case first.

**Note**: The material definition or density constant is used for each member, element, or solid to determine the self weight load.

1. Either:

   **On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group**

   **Tip**: This will add the load item to the currently selected load group selected in the program status bar.
or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select the **Selfweight > Selfweight Load** tab.

3. Select a global **Direction** for the selfweight.
   
   This is typically the “vertical” direction but can be any global direction.

4. Type a **Factor**.
   
   This is multiplier on the calculated weight of the structure objects. You do not need to use a load factor for ultimate strength design here, as that is typically applied to load combinations.
   
   **Note:** A positive value will act in the positive global direction, so you will typically use a negative value here for gravity loads (-1 being the most commonly used value).

5. Click **Add**.
   
   The load item is added to the selected load case.

You must assign the load items to structure objects.

**Tip:** As selfweight typically will act on all members, plates, etc., you can use the **Assign to View** option and click **Assign** to quickly assign this load item to all objects in the structure (assuming that you have all objects currently displayed in the view window).

**Related Links**

- [TR.32.9.1 Selfweight Loads](on page 2685)
- [Selfweight tab](on page 3007)

**M. To add a nodal load**

To add forces or moments that act at nodes, use the following procedure.

**Tip:** Nodal loads are also referred to as “Joint Loads” in the STAAD command reference.

1. Either:
   
   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.
   
   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
   
   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select the **Nodal Load > Node** tab.

3. Type the force and moment magnitudes acting in the global directions (with sign) applied at the same nodes.
   
   You can use any combination of forces and moments.

4. (Optional) To apply the load along an inclined coordinate system:
   
   a. Select the **Inclined Load?** option.
      
      The force and moment labels are updated with (’) to indicate that the loads and moments will act in the inclined axes.
b. Select the method of defining the reference point.
   - **Reference Node** - select an existing node number from the model. The global coordinates of that node are displayed as read-only.
   - **Absolute** - type the global coordinates of an arbitrary point to use as the reference.
   - **Relative** - type relative distances from the loaded joint to the reference point. These distances are measured along the global axes.

   **Tip**: To easily select a reference node with respect to an existing node, first use the **Reference Node** option to populate the fields with that node's coordinates. Then select the **Absolute** option to modify the values. For example, you can update the Y coordinate of an existing node to select a point directly above or below that node.

c. Type the global coordinates (Absolute) or distances (Relative) to the reference point.

5. Click **Add**.
   The load item is added to the selected load case.

**Related Links**
- Nodal Load tab (on page 3008)
- G.15.1 Joint Loads (on page 2337)
- TR.32.1 Joint Load Specification (on page 2651)
- Nodal Load tab (on page 3008)

M. To add a support displacement

To add a support displacement to a supported node, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   **Tip**: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.
   The **Add New Load Items** dialog opens.

2. Select the **Nodal Load > Support Displacement** tab.
3. Select the displacement **Direction**.
4. Either
   - type the **Displacement** distance for translations (i.e., Fx, Fy, or Fz directions).
   or
   - type the **Rotation** angle for rotations (i.e., Mx, My, or Mz directions).
5. Click **Add**.
   The load item is added to the selected load case.

**Related Links**
- G.15.7 Support Displacement Loads (on page 2342)
M. Member Load Items

M. To add a concentrated force or moment on members

To add either concentrated forces or members at a point along the length of a member, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select either:

   the **Member Load > Concentrated Force** tab

   or

   the **Member Load > Concentrated Moment** tab

3. Type the location along the length of the member to the concentrated load, \( d_1 \).
   This distance is measured from the start node of the member.

4. Type the lateral offset from the geometric centerline to the load, \( d_2 \).

5. Select the **Direction** option in which the load acts.
   Concentrated loads can be applied in the global or local coordinates.

6. Click **Add**.
   The load item is added to the selected load case.

Related Links

- **G.15.2 Member Load** (on page 2337)
- **TR.32.2 Member Load Specification** (on page 2653)
- **Member Load tab** (on page 3009)

M. To add a uniform load to members

To add a uniform force or moment along the full or partial member length, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select either:
   - the **Member Loads > Uniform Force** tab
   - or
   - the **Member Loads > Uniform Moment** tab

3. Type the magnitude of the distributed load (with sign), $W_1$.
4. Type the location along the length of the member to the start of the uniform load, $d_1$ and to the end of the uniform load, $d_2$.
   Both distances are measured from the start node of the member. If $d_2$ is zero, then the load will act to the end node of the member.
5. Type the lateral offset from the geometric centerline to the load, $d_3$.
6. Select the Direction option in which the load acts.
   Uniform loads can be applied in the global, local, or projected coordinates.
7. Click **Add**.
   The load item is added to the selected load case.

**Related Links**

- **G.15.2 Member Load** (on page 2337)
- **TR.32.2 Member Load Specification** (on page 2653)
- **Member Load tab** (on page 3009)

**M. To add a linear varying load to members**

To add linear varying force along the length of a member, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select :
   - the **Member Loads > Linear Varying** tab
   - or
   - the **Member Loads > Trapezoidal** tab

3. Select the load shape to apply:
   - **Load increases...**
   - select...  
     - **linearly along entire length of beam**
     - **the $W_1, W_2$ option on the Linear Varying** tab
Load increases... Select...
load increases from zero to peak at midspan and back to zero the W3 option on the Linear Varying tab
linearly along a partial section of the beam the Trapezoidal tab

4. Type the load values at the ends (W1, W2 option) or at midspan (W3 option) (with sign).
5. Select the Direction.
6. Click Add.
   The load item is added to the selected load case.

Related Links
• G.15.2 Member Load (on page 2337)
• TR.32.2 Member Load Specification (on page 2653)
• Member Load tab (on page 3009)

M. To add a prestress or post-tension load to members

To add a load due to prestressing or post-tensioning to members, use the following procedure.

1. Either:

   On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group
   
   Tip: This will add the load item to the currently selected load group selected in the program status bar.
   or
   In the Load & Definition dialog, select a primary load case in the Load Cases Details list and then click Add.
   The Add New Load Items dialog opens.

2. Select the loading Type:
   
   Prestress – prestress force is considered at the time of stressing and thus the force is transferred to the adjacent members or supports
   or
   Poststress - the prestress force is considered after the time of stress and thus the force is applied to the member itself (not transferred to the adjacent members or supports)

3. Type the prestress Force (in current units).

4. Type the Eccentricity Distances used to control the tendon profile at the Start, Middle, and End of the member.
   Eccentricities are measured in the local y axis (i.e., positive is “up” with respect to the local y axis). The cable profile is assumed to be parabolic (i.e, a “draped” tendon profile). Refer to G.15.5 Prestress and Poststress Member Load (on page 2340) for further details on how the tendon profile shape is calculated.

5. Click Add.
   The load item is added to the selected load case.

Related Links
• G.15.5 Prestress and Poststress Member Load (on page 2340)
• TR.32.5 Prestress Load Specification (on page 2678)
• Member Load tab (on page 3009)
M. To add fixed end member loads

To apply fixed end loads to ends of a beam, use the following procedure.

1. Either:

On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

**Tip:** This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add.**

The **Add New Load Items** dialog opens.

2. Select the **Member Loads > Fixed End** tab.

3. Type the load values to apply at the **Start Node** and **End Node.**

   These reference the ends of the member but the loads are applied directly to the member ends, not the joints. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member.

4. Click **Add.**

   The load item is added to the selected load case.

**Related Links**

- **G.15.4 Fixed End Member Load** (on page 2340)
- **TR.32.7 Fixed-End Load Specification** (on page 2683)
- **Member Load tab** (on page 3009)

M. Plate, Surface, Area, and Solid Load Items

M. To add pressure load on a plate

To add a uniform pressure load over an entire plate or a rectangular portion of a plate, use the following procedure.

1. Either:

On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

**Tip:** This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add.**

The **Add New Load Items** dialog opens.

2. Select either:

   - the **Plate Loads > Pressure on Full Plate** tab
   - or
   - the **Plate Loads > Partial Plate Pressure** tab
3. Type the magnitude of the uniform pressure (with sign), $W_1$.

4. For partial plate pressure loads, type the coordinates (with respect to the center of the element) to the first and second corners of the loaded rectangular area: $X_1, Y_1, X_2,$ and $Y_2$

![Figure 116: Coordinate values, x1,y1 & x2,y2, in the local coordinate system](image)

5. Select the **Direction** option in which the load acts.
   Pressure loads can act in any of the global directions: $GX$, $GY$, or $GZ$ as well as in the **Local Z** direction of the plate. Additionally, full pressure loads can act in the **Local Y** and **Local X** directions of the plate.

6. Click **Add**.
   The load item is added to the selected load case.

**Related Links**
- *G.15.8 Loading on Elements* (on page 2342)
- *Plate Loads tab* (on page 3013)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2657)

**M. To add an area load**
To add an loaded area which distributes pressure to beams forming a closed loop, use the following procedure.

*Note: The AREA LOAD command has been deprecated in favor of the ONEWAY LOAD or FLOOR LOAD commands.*

1. Either:
   - On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.
   
   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
   or
   - In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.
   The **Add New Load Items** dialog opens.

2. Type the magnitude of the **Pressure** in the current units.

3. Select the **Direction** in which the pressure acts.
- **Local Z** - parallel to the member local Z axis
- **GX** - parallel to the global X axis
- **GY** - parallel to the global Y axis
- **GZ** - parallel to the global Z axis

4. Click **Add**.
   The load item is added to the selected load case.

**Related Links**

- *G.15.3 Area, One-way, and Floor Loads* (on page 2338)
- *TR.32.4.1 Area Load Specification* (on page 2665)
- *Area Load tab* (on page 3012)

**M. To add a floor load or one-way load**

To generate a pressure load over a floor area using either two-way or one-way load distribution, use the following procedure.

1. Either:
   - On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group
     **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
   - or
   - In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.
     The **Add New Load Items** dialog opens.

2. Select the **Floor Load** tab.

3. Either:
   - select a range option –XRANGE, YRANGE, or ZRANGE– and then type the upper and lower bound distances (i.e., range) to define the members which define this loaded floor
   - or
   - select the **Group** option to apply the load to a named group and then select the Member Group name from the list
     **Tip:** This can be used to select a Composite Deck to assign a floor load to an entire deck. When decks are formed, this automatically creates a new group with the same name.

4. Type the **Pressure** in the current units.
5. Select the **Direction** in which the load acts.
6. (Optional) Select the **One Way Distribution** option to distribute the load towards the longer supporting members (i.e., along the shorter load path).
7. (Optional) When applying the load to a Group, you can check the **Inclined Floor** option to instruct the program that the load is applied to a set of members that form panels inclined to one of the global planes.
8. (Optional) You can report the panel and load data the program calculates:
   - a. Check the **Print to output file** to have data included in the STAAD Output and then select the data to include there.
b. Check the **Print to external text file** to have the data included in an external file and then select the data to include there.

9. Click **Add**.
   The load item is added to the selected load case.

**Related Links**
- **M. Composite Decks** (on page 693)
- **G.6.7 Composite Beams and Composite Decks** (on page 2328)
- **Composite Deck dialog** (on page 2916)
- **TR.20.7 Composite Decks** (on page 2473)
- **G.15.3 Area, One-way, and Floor Loads** (on page 2338)
- **Floor Load tab** (on page 3012)
- **TR.32.4.2 One-way Load Specification** (on page 2666)
- **TR.32.4.3 Floor Load Specification** (on page 2672)

**M. To add a surface selfweight load**

To add a selfweight load for surface elements so the weight of the surface elements is included in the analysis of the structure, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select the **Surface Loads > Selfweight Load** tab.

3. Select a global **Direction** for the selfweight.
   This is typically the “vertical” direction but can be any global direction.

4. Type a **Factor**.
   This is multiplier on the calculated weight of the structure objects. You do not need to use a load factor for ultimate strength design here, as that is typically applied to load combinations.

   **Note:** A positive value will act in the positive global direction, so you will typically use a negative value here for gravity loads (-1 being the most commonly used value).

5. Click **Add**.
   The load item is added to the selected load case.

   The SSELFwT command, along with direction and factor, are added to the load case.

   You must assign the load items to structure objects.

**Related Links**
- **TR.32.9.2 Surface Selfweight Load** (on page 2686)
M. To add a hydrostatic load to objects

To assign a hydrostatic load to members or plate elements, use the following procedure.

1. Either:
   - On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.
     Tip: This will add the load item to the currently selected load group selected in the program status bar.
   - or
     In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.
     The **Add New Load Items** dialog opens.

2. Select the appropriate tab:
   - **Object type**
     - **Members**
       - **Member Load > Hydrostatic**
     - **Plates**
       - **Plate Loads > Hydrostatic**

   Note: A message on the dialog indicates this option is disabled because no objects are selected.

3. Click **Select objects**.
   The **Selected Item(s)** dialog opens and the **Add New Load Items** dialog closes temporarily.

4. Use the appropriate selection tool to select all the objects to which you want to apply the hydrostatic load in the view window.
   All objects must be of the correct type to populate the list (e.g., only members can be added to the Member list, only plates can be added to the Plate list).

   Note: Hold the **<Ctrl>** to select multiple objects individually.
As objects are selected, their corresponding object id number is added to the object list.

5. Click **Done**.
   
The **Selected Item(s)** dialog closes and the **Add New Load Items** dialog re-opens. The selected objects are now displayed in the objects list.

6. Type values for the minimum and maximum global forces.

7. Select the direction in which the force acts and the global axis along which the program should interpolate loads.

8. Click **Add**.
   
The dialog closes and the hydrostatic load items are added to the selected load case.

Unlike most other loads, you do not need to assign these load items. The selection of the objects is used for assigning the load. The program will calculate the exact trapezoidal load pattern based on the other parameters.

**Related Links**

- *G.15.2 Member Load* (on page 2337)
- *TR.32.2 Member Load Specification* (on page 2653)
- *Member Load tab* (on page 3009)

**M. To calculate the structure frequency**

To calculate the frequency of the structure for a load case, use the following procedure.

Frequency calculation is made for a specific load case, so you must have at least one primary load case added to the input file.

This can be done by either the Rayleigh method or the eigenvalue extraction method. The latter can consider missing mass mode.

1. Either:
   
   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.

2. Select either:
   
   the **Frequency > Rayleigh Frequency** tab to use the approximation method
   
   or
   
   the **Frequency > Modal Calculation** tab to use the more exact eigenvalue extraction method

3. (Optional) For Modal Calculation, check the **Consider Missing Mass** option to include the missing mass procedure for steady state or harmonic analysis.

4. Click **Add**.
   
The frequency calculation is added to the load case.

**Related Links**

- *TR.34.1 Rayleigh Frequency Calculation* (on page 2790)
- *TR.34.2 Modal Calculation Command* (on page 2791)
M. Wind Loads

This section describes how to add wind loads to your STAAD.Pro model.

M. To add a wind load definition

To add a custom wind profile definition to your model, use the following procedure:

Wind load definitions describe the vertical wind profile that acts in a specific direction.

1. In the Load & Definition dialog, select the Definitions > Wind Load entry and then click Add.
   The Add new: Wind Definition dialog opens.
2. Type optional Comments and then click Add.
   The definition type is added to the Definitions > Wind Load entry in the Load & Definition dialog.
3. Either:
   - Repeat step 2 to add more wind definitions
   - or
   - click Close
4. Select the new wind definition type and click the Add.
   The Add New: Wind Definitions dialog opens. The dialog now displays the Intensity and Exposures tabs.
5. On the Intensity tab, select Custom in the Type drop-down list.
6. Enter the wind profile data in a specific direction in pressure and height above ground pairs.

   **Tip:** You can paste data copied from a spreadsheet.

   You can also parametrically calculate the wind profile based on ASCE 7. Refer to M. To add an ASCE 7 wind load definition (on page 836) for details.

   Alternatively, you can specify SNiP code wind parameters here. Refer to M. To add a SNiP wind load definition (on page 837) for details.
7. On the Exposure tab, type the Factor value.
   A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction.
8. Click Add.
9. (Optional) Repeat Steps 4, 7, and 8 to add additional exposure factors to this wind definition.
   You may add up to 99 exposure factors to a wind definition.

You must now apply this wind profile in a load case and specify the direction of the wind.

Related Links

- **G.16.3 Wind Load Generator** (on page 2345)
- **Create Wind Type Definition dialog** (on page 3038)
- **TR.31.3 Definition of Wind Load** (on page 2623)
M. To add an ASCE 7 wind load definition

To populate the wind intensity table with values calculated per ASCE 7, use the following procedure. These steps are performed when adding a wind load definition in the Add New: Wind Definitions dialog.

1. On the Intensity tab, select Custom in the Type drop-down list.
2. Click Calculate as per ASCE 7. The ASCE-7: Wind Load dialog opens.
3. On the Common tab, enter the general code and site information:
   a. Select the year (edition) of ASCE-7 code.
   b. Select the Building Classification Category.
   c. Type the Basic Wind Speed in the selected units.
   d. Select the Exposure Category.
   e. Select the Structure Type that best describes your structure.
   f. (Optional) If you need to Consider wind speed-up over Hills or Enscarpment, select the Yes option and then provide the rise type and dimensions
4. Click Apply.
   Note: The second tab of the dialog updates to reflect the Structure Type selection.
5. Select the second tab and enter the structure data:
   a. Type the structure dimensions.
   b. Select the structure cross-section type (where applicable).
   c. Select the orientation of members exposed to wind (where applicable).
   d. Select the shape of exposed structural members (where applicable).
   e. Type the Structure Natural Frequency.
   f. Type the Structure Damping Ratio.
   g. Select the Enclosure Classification (where applicable).
   h. (Optional) Select the Use Kzt or Use Kd options to manually type values for these coefficients. The calculated coefficients are otherwise used.
6. Select the Design Pressure tab.
   The height vs. intensity data calculated for the current input is displayed here.
7. (Optional) You can check the options for Use G and Use Cf to manually type values for these coefficients.
   The calculated coefficients are otherwise used.
8. Click OK.
   The dialog closes and the wind intensity at height data is added to the table on the Intensity tab.

Note: The input fill will contain a section on ASCE 7 wind load generation data. Though this is not read directly by the STAAD analysis engine, these values are stored to allow you to make changes to the wind load parameters.

You can now proceed with the wind load definitions using the calculated values per ASCE-7.

Related Links
- TR.31.3 Definition of Wind Load (on page 2623)
- ASCE 7 Wind Load dialog box (on page 3039)
To add a parametric wind load definition per the SNiP 2.01.07, SP 2.07.7, or SP 20.13330.2016 Loads and Actions building codes, use the following procedure.

**Tip:** Change units for convenience.

1. On the **Intensity** tab, select either of the following from the **Select Type** drop-down list:
   - SNiP 2.01.07-2011
   - SNiP 2.07.7-85
   The dialog updates to display the SNiP Parameter fields.
2. Type the **Wind Pressure**.
3. Select the **Terrain** type from the drop-down list.
4. Select the structure **Classification** from the drop-down list.
5. Select the **Region** from the drop-down list.
6. Type the **Delta** value.
7. Click **Add**.

**Related Links**

- **TR.31.3 Definition of Wind Load** (on page 2623)
- **TR.32.12.3 Generation of Wind Loads** (on page 2779)
- **Wind Load tab** (on page 3018)
- **TR.30.1 Cut-Off Frequency, Mode Shapes, or Time** (on page 2539)
- **TR.31.3 Definition of Wind Load** (on page 2623)
- **TR.32.12.3 Generation of Wind Loads** (on page 2779)
- **Add New Wind Definitions (data) dialog** (on page 3038)
- **Wind Load tab** (on page 3018)

**M. To assign exposure for joints**

To assign wind load definition exposure to joints, use the following procedure.

If your wind load definition has multiple exposure factors, you can specify the exact nodes for each factor this way.

This method assumes that you select the nodes first and then assign the exposure value to them. However, you can select the exposure method and then assign that to nodes using a different assignment method if more convenient.
1. Select the nodes which will all use a specific exposure ratio.
2. In the **Load & Definition** dialog, select that **Exposure** entry within a wind load definition.
3. Select the **Assign to Selected Nodes** method for assignment.
4. Click **Assign**.
   A message prompts you to confirm this assignment.
5. Click **Yes**.

The exposure ratio is assigned to these joints. When an exposure is selected, the assigned nodes are highlighted red in the view window.

You must now apply the wind load to the structure within a load case.

### M. To apply a wind load

To apply a defined wind load to your structure, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The Add New Load Items dialog opens.

2. Select the **Wind Load > Wind Load** tab.

   **Note:** If you have not previously added a wind load definition to the model, this tab is inactive. Refer to **M. To add a wind load definition** (on page 835) for details.

3. In the **Select Type** drop-down list, select the wind load definition for this load case.
4. Select the **Exposed Surface and Direction** of the load and then type the load **Factor** value.
5. (Optional) If the wind load definition is for a SNI code, specify the additional SNI Parameters:
   a. Select the option to **Apply Wind Load at Corners** if required.
   b. Select the configuration from the drop-down list.
   c. Type the wind pressure coefficient.
6. Specify the range of values over which the load applies. Type the along the global axis of how the wind load acts.
7. (Optional) Check the **Open Structure** option for truss, lattice, or other open structure types.
   This will apply the load only to the projected area of the members and joints, not to the bounded area of members or surfaces.
8. Click **Add**.

The program will automatically calculate the joint loads for structure that lies within the specified range. Selecting the wind load item in the **Load & Definition** dialog will highlight the loading members. The calculated surface exposure and the tributary areas will also be displayed for closed structures.
M. To apply a dynamic wind load per SP 20.13330.2016

To apply a SP 20.13330.2016 wind definition as a dynamic load to your structure, use the following procedure.

A modal analysis and at least one static load cases are required in order to use a dynamic wind load case.

In order to perform dynamic wind load generation in STAAD.Pro you must previously define a static load case. The static load case could be any primary static load case which is defined before the Russian dynamic load case, including a static wind loading per the same code. This static load case will provide static load vector to the dynamic wind load module.

As the Russian dynamic wind load component requires modal masses and eigen vectors to calculate the dynamic wind load component at nodes, modal analysis must be performed before the dynamic wind load definition. Therefore, you must also include a separate load case for modal analysis with reference mass defined before the load case. See G.17.3.2 Mass Modeling (on page 2365) for details. Alternatively, if the mass loads are not needed for use with other load cases (that is, a reference load case is not needed for other loads), then you may define the mass loads within the dynamic load case prior to the WIND LOAD command.

1. Either:

On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group.

Tip: This will add the load item to the currently selected load group selected in the program status bar.

or

In the Load & Definition dialog, select a primary load case in the Load Cases Details list and then click Add. The Add New Load Items dialog opens.

2. Select the Wind Load > Wind Load - Dynamic tab.

Note: If you have not previously added a SP 20.13330.2016 wind load definition to the model, this tab is inactive. Refer to M. To add a SNiP wind load definition (on page 837) for details.

3. In the Select Type drop-down list, select the wind load definition for this load case.

Only SP 20.13330.2016 wind definitions are present in this list.

4. Type the Width of building along wind dir. and Width of building across wind dir. values.

These are the width parallel and perpendicular to the wind direction, respectively.

5. Type the load Factor value.

6. (Optional) Check the Internal option to adjust the direction so that the wind acts as an internal pressure (i.e., suction on the surface).

7. (Optional) Check the Allow All Mode Shapes to include all mode shapes from the modal analysis.

Otherwise, only the first mode shape will be used.

8. Select the Static Wind Case from the drop-down list of primary load cases that precede this load case in the model.

9. Select the direction in which the wind is applied from the graphical plan view.

10. Click Add.

Related Links

- TR.31.3 Definition of Wind Load (on page 2623)
- TR.32.12.3 Generation of Wind Loads (on page 2779)
M. Seismic Loads

This section describes how to add seismic loads to your STAAD.Pro model.

M. To add a seismic load definition

To add seismic load parameters per a building code, use the following procedure.

Static equivalent seismic loads are applied based on a set of parameters used to define the load. The parameters vary for each building code.

2. On the Seismic Parameters tab, select the building code from the Type drop-down list.
3. (Optional) Check the Include Accidental Load option to consider accidental torsion.
   
   Note: Not applicable for the Canadian NRC 1995 code.
4. (Optional) For the Indian IS 1893-2002/2005 code:
   a. (Optional) Check the Include IS 1893 Part 4 option for industrial or stack-like structures.
   b. Click Generate to open the IS 1893 Seismic Parameters dialog, which provides a simpler method for entering the parameters with additional context for IS 1893 Part 1.
   c. Click Generate.
5. Type the seismic parameters in the table as required for the selected building code.
   A contextual note is displayed below the table for each parameter. Refer to TR.31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2544) for detailed descriptions of the parameters for each code.
6. Click Add.
   If you have not provided all the required parameters, a message opens to alert you of missing items. Once the minimum required parameters have been provided, the seismic definition is added to the Definitions > Seismic Definitions entry in the Load & Definition dialog. The Seismic Parameters tab is removed from the Add New: Seismic Parameters dialog.

You will need to add weight items to the load definition. You can do this through a series of loads, a reference load case, or a combination of both.

Related Links
- TR.31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2544)
- TR.31.2.5 Chinese Static Seismic per GB50011-2001 (on page 2563)
- Add New Seismic Definitions dialog (on page 3044)

M. To add weight items to a seismic load definition

To create structural weights for a seismic load definition, use the following procedure.

If you want to assign floor weights to a floor group, you must have at least one group of the type Floor previously defined to use this option.
Tip: If you want to use the same loads for both seismic weights and another load (e.g., the dead load case), you can use a Reference Load case instead of this procedure.

1. Select the tab in the **Add New: Seismic Definitions** dialog corresponding to the weight you want to add.

2. Specify the weight parameters:
   For the following weight types, enter the data as indicated:
   
   **Weight type** | **Data to enter**
   --- | ---
   Self | Type a **Self Weight Factor**
   Joint | Type a **Joint Weight**
   Member | Select the **Loading Type**. Type the **Weight** to use, the **Starting Distance**, and for uniform weights, the **Ending Distance**.
   Element | Type a uniform **Pressure**

   For Floor Weights, do the following:
   a. Type a uniform floor **Pressure**.
   b. Select if the load is applied over a **Range** or a floor **Group**.
   c. (Optional) Check the **One Way Distribution** option to use that load distribution to supporting members.
   
   You can select the default behavior or select a specific member towards which the load is directed.
   d. For the **Range** option, specify Y range limits and optionally X and Z range limits.
   e. For the **Group** option, select a named group of members.
   f. (Optional) For the **Group** option, select the **Inclined Floor** option if the group includes members forming a panel that is inclined from the global XZ plane.

   Refer to [TR.32.4.3 Floor Load Specification](on page 2672) for additional details on specifying floor loads.

3. Click **Add**.
4. Repeat steps 1 through 3 to add additional weights to your seismic load definition.
5. Click **Close**.

You must assign the weight items to the appropriate objects.

**M. To add weight by a reference load to a seismic load definition**

To use a previously defined reference load case for the weights in your seismic load definition, use the following procedure.

You must create a reference load case containing the load items to be used as weights in the seismic load. Refer to [M. To create a reference load](on page 873) for details.

This approach is preferable if you need to use the same weight tables for multiple loads or analysis.

1. Select the **Reference Load** tab in the **Add New: Seismic Definitions** dialog.
2. Either:
   
   select a reference load definition and then click **>** to add it to the Reference Load item
   
   or
   
   click **>>** to add all the available reference load definition to the Reference Load item.
3. (Optional) Type a **Factor** to apply to each reference load definition in the Reference Load item as needed.
4. Select the global direction along which the load is applied from the **Along** drop-down list.
5. Click Add.
6. Click Close.

Related Links
• TR.31.6 Defining Reference Load Types (on page 2642)

M. To add a seismic load

To add a seismic load to a primary load case, use the following procedure.

1. Either:
   On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group.
   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
   or
   In the Load & Definition dialog, select a primary load case in the Load Cases Details list and then click Add.
   The Add New Load Items dialog opens.
2. Select the Seismic Loads tab.
   **Note:** If you do not have a seismic load definition created, this tab is disabled.
3. Select the global Direction in which this seismic load acts.
4. (Optional) Type the scale Factor to apply to the load.
5. (Optional) Check the option to use Multiplying factor for Accidental Torsion Moment and type a Factor.
   This value is the ratio of the total building width and is taken from the center of mass for each level.
6. (Optional) Check the option to use Multiplying factor for Natural Torsion Moment and type a Factor.
   **Note:** This is only applicable for structures modeled with a rigid diaphragm.
7. Click Add.

Related Links
• G.16.2 Seismic Load Generator (on page 2343)
• TR.32.12.2.1 Generation of Seismic Loads (on page 2773)
• TR.32.12.2.2 Generation of IS:1893 Seismic Load (on page 2778)

M. Response Spectra

This section describes how to add response spectra to your STAAD.Pro model.

**Tip:** In STAAD.Pro, a response spectrum analysis is performed on the model when it contains response spectra load case. It is possible to use a linear, elastic analysis with a response spectrum load.
M. To add a generic response spectrum

To add a custom response spectrum (referred to as the “generic” method) to your model, use the following procedure.

**Note:** You must enter in spectra values using the dialog. To reference an external file containing spectra data, you must use the STAAD.Pro Editor.

1. Either:
   - On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.
     - **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
   - or
     - In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.
     - The **Add New Load Items** dialog opens.

2. Select the **Response Spectrum** tab.

3. Select **Custom** from the **Code** drop-down list.

4. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   **Note:** For the ASCE method, there are two optional parameters, FF1 and FF2, which can be checked and specified.

5. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.
   - The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an .acc file extension.

6. Specify the spectrum data:
   a. Select if the **Spectrum Type** uses **Acceleration** or **Displacement** values.
      - The spectrum data table updates accordingly.
   b. Type in the period and spectra data pairs in the table.
      - As you add pairs of data, the response spectrum plot is generated to help review the input.

7. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.
   - The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.

8. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.

9. Select the directions which spectrum is applied and, for each, type a factor (0-1.0).

10. (Optional) You can apply each of the following optional steps as needed:
    a. Select one of the **Signed Response Spectrum Results Options** if needed:
       - check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.
       - or
check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

**b.** To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

c. Type a **Scale** factor to apply to the spectra data.

d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.

e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.

**Related Links**
- TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688)
- Response Spectra tab (on page 3021)

**M. To add an IBC 2000 response spectrum**

To specify a response spectrum for seismic loading per the IBC 20005 code, use the following procedure.

The period and acceleration or displacement values for an IBC 2000 response spectrum can be generated by the program to use with a “custom” response spectrum load.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select the **Response Spectrum** tab.

3. Select **Custom** in the **Code** list.

4. Generate the IBC 2000 values for the table:

   **Note:** In order to use the generated spectrum values, you must leave the **Spectrum Type** selection as **Acceleration**.

   a. Click **Generate IBC Spectrum - 2000**.

   The **Spectrum Parameters** dialog opens.

   b. Specify the mapped acceleration values by either:

      select the **Zip** code from the drop-down list, click **Find Lat / Long**, and then click **Calculate S1 / SS**

   or
enter the **Latitude** and **Longitude** and then click **Calculate S1 / SS**

or

**enter the S1 and SS values directly**

c. **Select a Site Class** for the location.
   The Fa and Fv values for the site location are displayed.

d. **Type the Start, End, and Interval values for the period.**

e. **Click Generate Spectrum.**
   The Generated Spectrum: IBC 2000 dialog opens. The values are tabulated and the spectrum is plotted. You can use the settings on this dialog to review the data.

f. **Click Close.**
   The response spectrum values and plot is generated.

**Note:** If you select

5. **Select the Combination Method** to use to combine the responses from each mode into a total response.

   **Note:** For the ASCE method, there are two optional parameters, **FF1** and **FF2**, which can be checked and specified.

6. **(Optional)** Check the **Save** option to create a file containing the joint accelerations in ratios of gravity and in radians/sec².
   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an .acc file extension.

7. **Select if the Interpolation Type** is **Linear** or **Logarithmic**.
   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.

8. **Select the Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.

9. **Select the directions which spectrum is applied and for each, type a factor (0-1.0).**

10. **(Optional)** You can apply each of the following optional steps as needed:

    a. **Select one of the Signed Response Spectrum Results Options** if needed:
       check **Dominant** and type a **Mode No** to use for determining the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

       or

       check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

    b. **To generate individual modal response load cases,** check the **Generate load case[s] for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.
       If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

    c. **Type a Scale factor to apply to the spectra data.**

    d. **Check the Missing Mass option** to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.

    e. **If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead,** check the **ZPA** option and type a frequency.
If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.

   The new response spectrum is added to the currently selected load case.

**Related Links**
- *G.17.3.4 Response Spectrum* (on page 2371)
- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2688)
- *Spectrum Parameters dialog* (on page 3027)
- *Generated Spectrum dialog* (on page 3027)

**M. To add an IS 1893 response spectrum**

To specify a response spectrum for seismic loading per the IS 1893 code, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.

2. Select the **Response Spectrum** tab.

3. Select **IS-1893** in the **Code** list.

4. Specify the IS 1893 specific parameters:

   a. Select a **Subsoil class** for the location.

<table>
<thead>
<tr>
<th>Subsoil option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Custom</td>
<td>You must enter Period and Acceleration or Displacement value pairs in the table, based on the <strong>Interpolation Type</strong> selected. You can also select the <strong>Interpolation Type</strong> for values between data points.</td>
</tr>
<tr>
<td>Hard, Medium, or Soft Soil</td>
<td>Average response acceleration coefficient is calculated. You will not need to select <strong>Spectrum Type</strong> or <strong>Interpolation Type</strong> for these options.</td>
</tr>
</tbody>
</table>

   b. (Optional) Select the **Ignore mode(s) with mass participation (IGN)** option and then enter a percentage value.

   Local and torsional modes whose mass participation is less than this percent value are then considered negligible and therefore will be excluded.

   c. (Optional) Select the **Use Torsion** option and then enter the **Dynamic Eccentricity (DEC)** and **Accidental Eccentricity (ECC)** factors.

   The dynamic eccentricity factor is multiplied by the static eccentricity (i.e., the distance between the center of mass and center of rigidity) to get the dynamic eccentricity. This value should typically be 1.0 or higher.
The accidental eccentricity is used to calculate the value of accidental eccentricity. This value is typically 0.05 but may be increased to 0.10 for highly irregular buildings. This value can be positive or negative to accommodate clockwise or counter-clockwise rotation.

The response spectrum plot is generated as these parameters are specified.

5. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   **Note:** For the ASCE method, there are two optional parameters, FF1 and FF2, which can be checked and specified.

6. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec².

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an .acc file extension.

7. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.

8. Select the **Damping Type** to use and, if explicit Damping is selected, type the damping ratio to be used for all modes.

9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).

10. (Optional) You can apply each of the following optional steps as needed:

    a. Select one of the **Signed Response Spectrum Results Options** if needed:

       check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

       or

       check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

    b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

       If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

    c. Type a **Scale** factor to apply to the spectra data.

    d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.

    e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

       If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.

    The new response spectrum is added to the currently selected load case.

**Related Links**

- [TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002](#) (on page 2721)
M. To add an IBC 2006 response spectrum

To specify a response spectrum for seismic loading per the IBC 2006 / ASCE 7-05 codes, use the following procedure.

1. Either:

On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group.

Tip: This will add the load item to the currently selected load group selected in the program status bar.

or

In the Load & Definition dialog, select a primary load case in the Load Cases Details list and then click Add. The Add New Load Items dialog opens.

2. Select the Response Spectrum tab.

3. Select IBC 2006/ASCE 7-05 in the Code list.

4. Specify the IBC 2006 / ASCE 7-05 specific parameters:
   a. Specify the mapped acceleration values by either:
      - typing the Zip code (location and values are populated)
      - delete the Zip code value and type Latitude and Longitude (values are populated)
      - delete the Latitude value and type the Ss and S1 values directly
   b. Type the Long Period (TL) value for the structure.
   c. Type the Fa and Fv values for the soil conditions at the location.
   d. Select a Site class (SCL) for the location.

The response spectrum plot is generated as these parameters are specified.

5. Select the Combination Method to use to combine the responses from each mode into a total response.

6. (Optional) Check the Save option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an .acc file extension.

7. Select if the Interpolation Type is Linear or Logarithmic.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.

8. Select the Damping Type to use and, if explicit Damping is selected, type the damping ratio to be used for all modes.

9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).

10. (Optional) You can apply each of the following optional steps as needed:
a. Select one of the **Signed Response Spectrum Results Options** if needed:

check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

or

check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

c. Type a **Scale** factor to apply to the spectra data.

d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.

e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.

The new response spectrum is added to the currently selected load case.

**Related Links**

- [TR.32.10.1.8 Response Spectrum Specification per IBC 2006](on page 2743)

**M. To add an EC8 response spectrum**

Floor height is determined by the program as a joint where one or more beams frame in a column.

Eurocode 8 requires the use of cracked section stiffness when considering concrete buildings, which however is lacking in the current analysis engine. This can be overcome by using a section reduction factor as suggested in the code. STAAD.Pro has a "section reduction factor" that can be used for this purpose (i.e., the **MEMBER CRACKED** command). Refer to [TR.20.10 Member Property Reduction Factors](on page 2485) for details on this command.

**Note:** Refer to [TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004](on page 2715) for additional information on using the response spectrum specifications per Eurocode 8.

1. Either:

On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

**Tip:** This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select the **Response Spectrum** tab.

3. Select **EURO (EC8) - 2004** in the **Code** list.
4. Specify the EC8-2004 specific parameters:
   a. In the Spectrum table group, type the **Design Ground Acc(eleration) factor**.
   b. Type the **Behaviour Factor**.
   c. Select the **Subsoil Class** from the drop-down list.
   d. In the **Load Type** group, select if the load is in the **Elastic** or **Design** range and then select if the load is **Type I** or **Type II**.

   The response spectrum plot is generated as these parameters are specified.

5. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   **Note:** For the **ASCE** method, there are two optional parameters, **FF1** and **FF2**, which can be checked and specified.

6. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec².

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an .acc file extension.

7. Select if the **Interpolation Type** is Linear or Logarithmic.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.

8. Select the **Damping Type** to use and, if explicit Damping is selected, type the damping ratio to be used for all modes.

9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).

10. (Optional) You can apply each of the following optional steps as needed:

    a. Select one of the **Signed Response Spectrum Results Options** if needed:

        check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

        or

        check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

    b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

        If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

    c. Type a **Scale** factor to apply to the spectra data.

    d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.

    e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

        If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.

    The new response spectrum is added to the currently selected load case.
This response spectrum may be used with a linear, elastic analysis, geometric nonlinear analysis, or a pushover analysis for review of Eurocode 8 seismic requirements in the Earthquake workflow (on page 2240).

Related Links
• TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2715)
• P. Using the Earthquake Workflow (on page 2240)

M. Snow Loads

This section describes how to add snow loads to your STAAD.Pro model.

M. To add an ASCE 7-02 snow load

Loads and definitions are added from the Load & Definition dialog in the Loading page.

Snow load definitions are applied to floor groups. You must have at least group of the type Floor previously defined.

Tip: You may want to change your input units to make entering snow loads convenient.

1. In the Load & Definition dialog, select the Definitions > Snow Definition entry and click then Add.
   The Add New: Snow Definition dialog opens.
2. Enter the general snow parameters and click the Add button.
   The newly created snow load definition will appear in the Definitions > Snow Definition section in the Load & Definition dialog.
3. Either:
   Repeat step 2 to add a second snow load definition.
   or
   Click Close to stop adding snow load definitions.
4. (Optional) Create a new primary load case for use with the snow load.
5. In the Load & Definition dialog, select the primary load case which will include the snow load.
6. Click Add.
   The Add New: Load Items dialog opens.
7. Select the Snow Load tab.

   Note: If no floor groups have been defined, this page will be inactive and a warning message is displayed in the dialog.
8. Select the Floor Group and select the Snow Load parameters.
9. Click Add.
   The snow load is added to the primary load case.

Related Links
• Add New Snow Definition dialog (on page 3043)
• TR.31.5 Definition of Snow Load (on page 2641)
• G.16.4 Snow Load (on page 2345)
• Snow Load tab (on page 3021)
• TR.32.13 Generation of Snow Loads (on page 2785)
M. Notional Loads

Notional loads are nominal lateral loads used in a direct analysis, such as specified in AISC 360.

M. To define direct analysis parameters

To define the member parameters used in an AISC 360 direct analysis, use the following procedure.

1. On the Loading ribbon tab, select the Direct Analysis tool in the Define Load Systems group.
   The Create New Load Items dialog open.
2. Specify the initial $\tau_b$ value:
   Members whose flexural stiffness are considered to contribute to the lateral stiffness of the structure will have their flexural stiffness (EI) reduced by $0.8 \times \tau_b$.
   a. Select the FLEX Parameter tab.
   b. Type a FLEX value.
   c. Click Add.
3. Specify the yield strength value:
   a. Select the FYLD Parameter tab.
   b. Type a FYLD value.
   c. Click Add.
4. (Optional) Identify any axial members which contribute to the lateral stiffness of the structure:
   a. Select the AXIAL Parameter tab.
   b. Click Add.
5. Click Close.
6. In the Load & Definition dialog, expand the Definitions > Direct Analysis Definition group.
7. Assign each of the FLEX, FYLD, and AXIAL parameters listed to the corresponding members in the model.

You must add one or more load cases with notional loads to your model for a direct analysis.

Related Links

- G.17.2.1.4 AISC 360 Direct Analysis (on page 2352)
- TR.31.7 Definition of Direct Analysis Members (on page 2643)
- Add New Direct Analysis Definition dialog (on page 3047)

M. To add a notional load case

To add notional loads to a load case for use with direct analysis, use the following procedure.

Notional loads are lateral loads which are defined as a small percentage of gravity loads. Therefore, you must define your gravity loads in a separate reference load case (recommended) or primary load case before adding notional loads.

The notional loads are calculated as lateral loads applied to the joints in the selected global direct.

1. Either:

   On the Loading ribbon tab, select the Load Items tool in the Loading Specifications group
Tip: This will add the load item to the currently selected load group selected in the program status bar.

or

In the Load & Definition dialog, select a primary load case in the Load Cases Details list and then click Add. The Add New Load Items dialog opens.

2. Select the Repeat Load > Notional Load tab.

Two load lists are displayed. The first is a list of primary load cases which occur before the current primary load case and the second is a list of reference load definitions.

3. Either:

   select a load case or reference load case on the right and click > to add it to the notional load
   or
   click either >> to add either all of the load cases or reference loads to the notional load

4. Type a Factor to use for the ratio of lateral to gravity loads for each load case added to the notional load.

   Typically, codes recommend 0.2 to 0.3 % (0.002 to 0.003). The default value will be the Notional Load Factor specified in the Direct Analysis Definition.

5. Select the global Direction for each load case added to the notional load.

6. Click Add.

Related Links

- G.17.2.1.4 AISC 360 Direct Analysis (on page 2352)
- TR.32.14 Notional Loads (on page 2785)
- Repeat Load tab (on page 3028)

M. Moving Loads

This section describes how to generate basic moving loads to your STAAD.Pro model.

Moving loads generated by the moving load generator in STAAD.Pro are only applied to beams. Plates and surfaces cannot be loaded using these loads.

For bridge decks which are skewed with respect to the global axes, the load generation may not yield the most satisfactory results. In such cases, the M. Bridge Deck workflow (on page 883) is recommended is recommended. The Bridge Deck workflow works on the influence line/influence surface method, and is considerably superior to the moving load generator. It also has the advantage of being able to calculate the critical load positions on decks modeled using plate elements.

Related Links

- TR.31.1 Definition of Moving Load System (on page 2541)
- V. Moving Load Generator (on page 3546)

M. To define a vehicle for loading

In order to generate a set of static loads, due to the movement of the vehicle or load on the structure, there are multiple steps involved.

Note: This procedure is used to generate a set of static loads at intervals on a structure. The Bridge Deck workflow (on page 883) can also be used to automatically place loads at locations to achieve maximum or minimum actions on a bridge deck.
1. Either:
   - on the **Loading** ribbon tab, select the **Moving Loads** tool in the **Loading** group

   or

   Select **Vehicle Definitions** under **Definitions** in the **Load & Definition** dialog and then click **Add**.
   The **Add New Vehicle Definition** dialog opens.

2. Specify a **Vehicle Type Ref** number.
   This identification number is used to refer to this vehicle load definition in the moving load generator.

3. Either:
   - define a custom vehicle using the **Define load** tab
   - select a standard AASHTO vehicle definition using the **AASHTO Spec** tab
   - select an external file containing vehicle data using the **File Input** tab

4. Click **Add**.

**Related Links**
- *G.16.1 Moving Load Generator* (on page 2343)
- *Add New Vehicle Definitions dialog* (on page 3048)
- *TR.31.1 Definition of Moving Load System* (on page 2541)

**M. To generate moving load cases**

To create a generation of load cases, use the following procedure.

1. In the **Load & Definition** dialog, select the **Load Case Details** entry and then click **Add**.
   The **Add New: Load Cases** dialog opens.
2. Select the **Load Generation** tab.
3. Type the **No. of Loads** to be generated.
   This is the total number of load positions that will be generated. You will later define the starting location and the increment distance between each position.
4. (Optional) Select the **Predefined Loads to be Added** from the drop-down list.
5. Click **Add**.

The moving load generation case is added to the **Load Case Details** entry in the **Load & Definition** dialog. These cases are marked with a [G] to indicate that they are generations of load cases. The load case number range is given in the title.

**Related Links**
- *G.16.1 Moving Load Generator* (on page 2343)
- *Load Generation dialog* (on page 3031)
- *TR.32.12.1 Generation of Moving Loads* (on page 2771)
M. To add vehicles to the load generation

To add a vehicle definition to the moving load generation, use the following procedure.

1. Select a moving load generation case in the Load Case Details entry in the Load & Definition dialog and then click Add.
   The Add New: Load Cases dialog opens.
2. Select the vehicle definition in the Type drop-down list.
3. Type the coordinates for the Initial Position of the Load.
   This is where the reference load (i.e., the first specified concentrated load in the moving load system) is placed on the structure for the first load case in the generation.
4. Type the Load Increment values in the planar directions to move the load for each subsequent load case.
5. Click Add.
6. (Optional) Repeat steps 2 through 5 to add additional moving loads to the load generation case.
7. Click Close.

Related Links
• (Moving) Load Generation Type dialog (on page 2999)
• TR.32.12.1 Generation of Moving Loads (on page 2771)

M. Time History Loads

This section describes how to add time history loads to your STAAD.Pro model.

M. To define a time history type from tabular data

To use a set of time value pairs to define a time history load, use the following procedure.

1. On the Loading ribbon tab, select the Time History > Forcing Function tool in the Dynamic Specifications group.

   The Add New: Time History Definitions dialog opens.
2. Type an Integration Time Step value in seconds.
3. (Optional) Select the Consider Missing Mass Mode option if necessary.
4. (Optional) Select the Save option to generate a file with time history data.
5. Select the Loading Type for your data:
   Acceleration
   or
   Force
   or
   Moment
6. Type the **Define Time vs** load data values in the table. As you add data pairs, the chart below is populated.

7. Click **Add**.

**Note:** You must define time history parameters to complete the time history load definition.

**Related Links**
- *G.17.3.5 Response Time History* (on page 2373)
- *TR.31.4 Definition of Time History Load* (on page 2630)
- *Add New Time History Definitions dialog* (on page 3056)

**M. To define a time history type from a function**

To use a parametric function to define a time history load, use the following procedure.

1. On the **Loading** ribbon tab, select the **Time History > Forcing Function** tool in the **Dynamic Specifications** group.

   The **Add New: Time History Definitions** dialog opens.

2. Type an **Integration Time Step** value in seconds.
3. (Optional) Select the **Consider Missing Mass Mode** option if necessary.
4. (Optional) Select the **Save** option to generate a file with time history data.
5. Select the **Loading Type** for your data:
   - Acceleration
   - Force
   - Moment

6. Select the **Harmonic Function** option in the Function Options group.
7. Select the harmonic function to use: **SINE** or **COSINE**.
8. Select one of the options for specifying wave length:
   - select **Frequency** and enter a cyclic frequency in cycles/sec.
   - select **RPM** and enter the revolutions per minute
9. Specify the harmonic wave parameters:
   - **a.** Enter the **Amplitude** in the units indicated.
     These correspond to the current input units and the **Loading Type** selection.
   - **b.** (Optional) Enter the **Phase** angle in degrees.
   - **c.** Enter the number of **Cycles** of loading.
10. Specify the number of time steps per cycle by either:
    - select the **Step** option and enter the time step of the loading.
or

select the SubDiv option and enter a number of subdivisions per quarter cycle (i.e., from zero to peak amplitude along the sine or cosine wave).

11. Click Add.

**Note:** You must define time history parameters to complete the time history load definition.

**Related Links**
- G.17.3.5 Response Time History (on page 2373)
- TR.31.4 Definition of Time History Load (on page 2630)
- Add New Time History Definitions dialog (on page 3056)

### M. To define a time history type by spectrum

To define a time history type using spectral parameters, use the following procedure.

1. On the **Loading** ribbon tab, select the **Time History** > **Forcing Function** tool in the **Dynamic Specifications** group.

   The **Add New: Time History Definitions** dialog opens.

2. Type an **Integration Time Step** value in seconds.

3. (Optional) Select the **Consider Missing Mass Mode** option if necessary.

4. (Optional) Select the **Save** option to generate a file with time history data.

5. Select the **Loading Type** for your data:
   - Acceleration
   - Force
   - Moment

6. Select the **Spectrum** option in the Function Options group.

7. Specify the time data:
   a. Enter the maximum time (in seconds) in the generated time history in the **Tmax** field.
   b. Enter the time step (in seconds) in the **DeltaT** field.
   c. Enter the time at the end of rising acceleration in the **T1** field.
   d. Enter the time at the end of the steady acceleration in the **T2** field.
   e. Enter the time at the end of the acceleration decay in the **T3** field.

8. Enter the damping ratio in the **Damp** field.

9. Specify the options:
   a. (Optional) Type a **Random Seed** value.

   Enter a positive integer (1 to 2147483647) to be used as a unique random number generation "seed." A unique time history will be produced for each seed value. Change this value when you want to produce a
“different (from the time history generated with the prior seed value)” but statistically equivalent time history.

b. Enter the No of digitized Freq.  
   This is the number of equally spaced frequencies at which the input shock spectrum is re-digitized (by interpolation).

c. Enter the No. of Iteration which will be used to perfect the computed time history.

10. Click Add.

For response spectra time history, there are some additional output parameters that must be input via the STAAD.Pro Editor if used.

**Note:** You must define time history parameters to complete the time history load definition.

**Related Links**
- [G.17.3.5 Response Time History](#)
- [TR.31.4 Definition of Time History Load](#)
- [Add New Time History Definitions dialog](#)

### M. To generate output for time history spectrum

To generate some additional output data for spectrum input for a time history definition, use the following procedure.

Create a structure with a time history definition using the Spectrum option.

These optional commands cannot be entered via the graphical user interface. They are required to be added using the STAAD.Pro Editor.

1. On the **Utilities** ribbon tab, select the **Command File Editor** tool in the **Utilities** group.

   The STAAD.Pro Editor window opens with the current STAAD input file.

2. (Optional) To output time history data: after the DEFINE TIME HISTORY command, in the SPECTRUM options, add the command THPRINT $f_{18}$.

   Where $f_{18}$ directs the program to output either the beginning and last 54 data points ($f_{18} = 1$) or the entire curve ($f_{18} = 2$) or a select number of beginning and last data points ($f_{18} \geq 10$).

   The spectrum input parameters will be included in the STAAD Output (.anl) file along with the Time History Output (limited to the $f_{18}$ value specified).

3. (Optional) To generate spectrum output for a time history: after the DEFINE TIME HISTORY command, in the SPECTRUM options, add the command SPRINT $f_{19}$.

   Where $f_{19}$ represents an integer value after the SPRINT command to instruct the program to only output the beginning and last number of values equal to this integer.

   A summary of the Spectrum input and the curve points will be included in the STAAD Output (.anl) file.

4. Save the command input file and then exit the STAAD.Pro Editor.

**Related Links**
- [TR.31.4 Definition of Time History Load](#)
M. To use frequency-spectra pairs in a time history load

To instruct the program to use frequency-spectra pairs in lieu of period-spectra pairs for the time history spectrum input, use the following procedure.

Create a structure with a time history definition using the Spectrum option.

These optional commands cannot be entered via the graphical user interface. They are required to be added using the STAAD.Pro Editor.

1. On the **Utilities** ribbon tab, select the **Command File Editor** tool in the **Utilities** group.

   ![Command File Editor](image)

   The STAAD.Pro Editor window opens with the current STAAD input file.

2. After the `DEFINE TIME HISTORY` command, in the `SPECTRUM` options, add the command `FREQ`.

3. Save the command input file and then exit the STAAD.Pro Editor.

**Related Links**

- *TR.31.4 Definition of Time History Load* (on page 2630)

M. To define a time history type by external file

To define a time history type using data in an external text file, use the following procedure.

You must have a separate text file containing pairs of times and values (acceleration, force, or moment). The file extension is not important, but the values must be in plain text.

1. On the **Loading** ribbon tab, select the **Time History > Forcing Function** tool in the **Dynamic Specifications** group.

   ![Time History Forcing Function](image)

   The **Add New: Time History Definitions** dialog opens.

2. Type an **Integration Time Step** value in seconds.

3. (Optional) Select the **Consider Missing Mass Mode** option if necessary.

4. (Optional) Select the **Save** option to generate a file with time history data.

5. Select the **Loading Type** for your data:

   - Acceleration
   - Force
   - Moment

6. Select the **From External File** option in the Function Options group.

7. Type the **File Name** (with or without file extension).

8. Click **Add**.
An example of an external data file:

<table>
<thead>
<tr>
<th>0.0</th>
<th>1.0</th>
<th>1.0</th>
<th>1.2</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>1.8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.0</td>
<td>2.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.0</td>
<td>2.6</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

If you need to specify a delta time spacing used for the external file data, this must be done using the STAAD.Pro Editor.

**Note:** You must define time history parameters to complete the time history load definition.

### Related Links
- [G.17.3.5 Response Time History](#) (on page 2373)
- [TR.31.4 Definition of Time History Load](#) (on page 2630)
- [Add New Time History Definitions dialog](#) (on page 3056)

### M. To define time history parameters

To define arrival times and damping used for time history loads, use the following procedure.

You must first define a time history type by one of the previously described methods.

One set of time history parameters can be defined for your model for all time history load types.

1. On the **Loading ribbon** tab, select the **Time History > Parameters** tool in the **Dynamic Specifications** group.

   The **Define Param** dialog opens.

2. **(Optional) Check the Time Step option and type the time step value.**
   
   This is the solution time step used in step-by-step integration of the uncoupled equations. If not selected, the default value of 0.0013888 seconds is used.

3. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.

4. **Type in one or more Arrival Times for the dynamic loads.**
   
   Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion).

5. **Click Add.**

The time history load parameters are added to the **Time History Definitions** in the **Load & Definition** dialog.

In order to use a time history load definition, you must add a time history load item to a primary load case.

### Related Links
- [G.17.3.5 Response Time History](#) (on page 2373)
- [Define (Time History) Parameters dialog](#) (on page 3058)
- [TR.31.4 Definition of Time History Load](#) (on page 2630)
M. To add a time history load

To add a time history load to the current primary load case, use the following procedure.

Select the current load case in the program status bar.

1. Either:
   - On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.
     
     **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
   - or
     - In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select the **Time History** tab.
3. Select the **Load Type**:
   - **Time Load** for a forcing function that applies forces or moments to joints
   - or
   - **Ground Motion** is applied global to all supports as an acceleration type
4. Select Direction in which the load is applied.
   - Time Load types are applied in FX, FY, or FZ for forces and MX, MY, or MZ for moments. Ground motion types are applied in one of the three global directions.
5. Select the arrival time for this load from the arrival times in the time history definition parameters.
6. Select the load definition type from the Defined Types drop-down list.
   - The list includes the definition number and function type.
7. (Optional) Select the results type from the Response Types drop-down list.
8. (Optional) Type a Force Amplitude Factor to multiply for the forces, moments, or accelerations at the joints.
9. Click **Add**.

You must assign the time history load to the joints to which it applies.

**Related Links**
- **G.17.3.5 Response Time History** (on page 2373)
- **Time History tab** (on page 3017)
- **TR.32.10.2 Time Varying Load for Response History Analysis** (on page 2767)

M. Pushover Loads

This section details create a pushover loads for use with a pushover analysis in STAAD.Pro.

**Note:** An advanced analysis module license is required to use a pushover analysis.

**Related Links**
- **Perform Pushover Analysis tab** (on page 3075)
- **A. To specify a pushover analysis** (on page 933)
M. To define general pushover data

1. On the Loading ribbon tab, select the Pushover tool in the Define Loading Systems group. The Create New Definitions / Load Cases / Load Items dialog opens with only the Pushover tab displayed.
2. On the Define Input tab (on page 3049), select the General Input Parameters option.
3. (Optional) Select the Type of Frame
4. (Optional) Select Include Effect Geometric Non-Linearity Effect

Related Links
- Define Input tab (on page 3049)
- G.17.4.2.1.1 Define Steel Moment and Braced Frames (on page 2386)
- TR.37.7.2.1 Type of Frame (on page 2819)
- Define Input tab (on page 3049)
- TR.37.7.2.4 Consideration of Geometric Nonlinearity Effect (on page 2820)
- Define Input tab (on page 3049)
- TR.37.7.2.5 KG Matrix Iteration (on page 2821)
- Define Input tab (on page 3049)
- TR.37.7.2.6 Maximum number of Analysis cycle (on page 2821)
- Define Input tab (on page 3049)
- TR.37.7.2.8 Save Output Results for Multiple Steps (on page 2822)
- Define Input tab (on page 3049)
- TR.37.7.2.7 Print Output Result (on page 2822)
- Define Input tab (on page 3049)
- TR.37.7.7 End Pushover Data (on page 2828)

M. To define member specific pushover data

Use the following optional procedure to specify parameters for individual members.

1. On the Loading ribbon tab, select the Pushover tool in the Define Loading Systems group. The Create New Definitions / Load Cases / Load Items dialog opens with only the Pushover tab displayed.
2. On the Define Input tab, select the Member Specific Parameters option.
3. (Optional) Set the Expected Yield Stress
4. (Optional) Set the Effective Length Factor for Member (Y/Z)
5. Click Add.
6. Once any other parameters have been added, click Close. Entries for the member parameters are added in the Loads & Definitions dialog in the Definitions > Pushover Definitions section.
7. Select one of the member parameters and click Assign.

Note: Refer to Graphical Environment help for using the various assignment methods in the Loads & Definitions dialog.

Related Links
- Define Input tab (on page 3049)
- TR.37.7.2.2 Expected Yield Stress (on page 2819)
M. To manually define and assign hinges

Use the following optional procedure to define parameters and assign hinges manually.

**Note:** If the pushover analysis will make use of the built-in FEMA hinge properties for all members,

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group. The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. Select the **Define Hinge Property** tab.
3. Select the **Hinge Type**.
   Types **FEMA** or **Ignore** do not require any additional parameters.
4. (Optional) For user-defined hinge types only:
   a. Select an existing **Type ID** or select **New Type** to define a new hinge type.
   b. Specify a unique integer for the **Type Identifier**.
   c. Specify coordinate values for points C, D, and E in the **Load Deformation Curve Points** and IO, LS, and CP values in the **Acceptance Criteria**.
      These values are can be determined using Tables 5-6 and 5-7 in FEMA 356.
   d. Specify values for **Yield Moment (YR)** and **Yield Rotation (YR)**.
      These values are calculated per Section 5.5.2.2.2 of FEMA 365 ([See “Frame element hinge properties”](on page 2382)).
5. Click **Add**.
6. Once any other parameters have been added, click **Close**.
   Entries for the hinge types are added in the **Loads & Definitions** dialog in the **Definitions > Pushover Definitions** section.
7. Select one of the hinge types and click **Assign**.

**Note:** Refer to Graphical Environment help for using the various assignment methods in the Loads & Definitions dialog.

**Related Links**

- **Define Hinge Property tab** (on page 3053)
- **G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria** (on page 2388)
- **G.17.4.1.4 Types of Nonlinearity** (on page 2381)
- **G.17.4.1.6 Frame element hinge properties** (on page 2382)
- **TR.37.7.5.1 User-Defined Hinge Property** (on page 2826)
- **TR.37.7.5.2 Assignment of Hinge Property to the Members** (on page 2827)
- **Define Hinge Property tab** (on page 3053)

M. To define pushover spectral data

Use the following procedure to define the seismic hazard per FEMA 356.

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group. The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. Select the **Define Spectral Details** tab.
3. Specify up to four **Critical Damping** values for the 1st through 4th spectra. At least one is required, which has a default value of 5.0%.
4. (Optional) Select the **Site Category**
   Site Class D is used by default.
5. Specify the Mapped Spectral Acceleration **At Short Period**, $S_s$, to be used per Table 1-4 of FEMA 356.
6. Specify the Mapped Spectral Acceleration **At One-Second Period**, $S_1$, to be used per Table 1-5 of FEMA 356.
7. Click **Add**.

**Related Links**
- [G.17.4.2.1.7 Define Input for Demand Spectrum](on page 2390)
- [TR.37.7.6 Define Spectral Parameters](on page 2828)
- [Define Spectrum Details tab](on page 3053)

M. To add a pushover loading

Use the following procedure to define the pushover loading to be used in the pushover analysis.

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group. The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. Select the **Define Loading Pattern** tab.
3. (Optional) Select **User Defined**
   If the default Auto load pattern is used, the program will internally compute the gravity loads.
4. (Optional) Set the **Method for Lateral Load Calculation**
   See "Program Defined Push Load Distribution Pattern" (on page 2823) for additional information on each method.
5. (Optional) Set the **Total Base Shear to be Distributed Direction Total Base Shear**
6. (Optional) Specify a **Number of Push Load Steps**
7. Click **Add**.

**Tip:** Only one pushover loading definition may be used for a model. If a different pushover loading definition is required, either change the individual parameters or delete them and create a new definition.

**Related Links**
- [TR.37.7.3.2 Total Base Shear to be Distributed](on page 2824)
- [Define Loading Pattern tab](on page 3051)
- [TR.37.7.3.3 Number of Push Load Steps](on page 2824)
- [Define Loading Pattern tab](on page 3051)
- [TR.37.7.3.1 Program Defined Push Load Distribution Pattern](on page 2823)
- [Define Loading Pattern tab](on page 3051)
- [G.17.4.2.1.3 Define Lateral (Push) Loading](on page 2387)
- [G.17.4.1.8 Lateral Load Distribution](on page 2385)
- [TR.37.7.3 Define Loading Pattern](on page 2823)
- [TR.37.7.8 Pushover Loading Input](on page 2829)
- [Define Loading Pattern tab](on page 3051)
M. To define solution control

Use the following procedure to define at least one solution control method for the pushover analysis.

1. On the Loading ribbon tab, select the Pushover tool in the Define Loading Systems group. The Create New Definitions / Load Cases / Load Items dialog opens with only the Pushover tab displayed.
2. Select the Define Solution Control tab.
3. (Optional) Set the Push Up to Defined Base Shear Direction Defined Base Shear
4. (Optional) Set the Push Up to Defined Displacement at Control Joint Direction Joint Displacement Value Joint Number
5. Click Add.

Related Links
- G.17.4.2.1.6 Define Pushover Analysis Solution Control (on page 2390)
- TR.37.7.4 Define Solution Control (on page 2825)
- Define Solution Control tab (on page 3055)

M. To use starting vectors with load-dependant Ritz vectors

You must set the eigen method to use load-dependant Ritz (LDR) vectors in order to specify the starting vectors for that method. On the Analysis and Design ribbon tab, select the Miscellaneous Commands > Set Eigen Method tool in the Analysis Data group. Then select the Load Dependant Ritz Vectors (LDR) option and click OK.

Note: This method is used to specify starting mass loads only. In order to specify starting load from reference load cases, you must directly edit the STAAD input file.

1. Select the Loading page in the Analytical Model workflow.
2. Select Starting Load Definition under Definitions in the Load & Definition dialog and then click Add. The Add New Define Starting Mass Load dialog opens.
3. Select the directions you want to use for the starting mass load vectors.
4. (Optional) For each direction selected, enter the number of vectors to be extracted corresponding to that load.
5. Click OK.

Related Links
- TR.31.9 Defining Starting Load (on page 2648)
- G.17.3.1 Solution of the Eigenproblem (on page 2363)
- Add New Define Starting Mass Load dialog (on page 3000)
- TR.31.9 Defining Starting Load (on page 2648)

M. Load Combinations

This section describes how to add load combinations to your STAAD.Pro results.
Note: Load combinations of primary load cases are algebraic combination of analysis results. Therefore, they are not applicable to nonlinear analyses.

Load Combinations with Loads from STAAD.Pro Physical Modeler

When you are creating load combinations using loads from STAAD.Pro Physical Modeler, there are two approaches:

1. Creating primary loads for each reference load generated from STAAD.Pro Physical Modeler. This will combine the analysis results of primary load cases and allow you to include the STAAD.Pro Physical Modeler loads in those results.
2. Using repeat load combinations. These load combinations will use the reference load cases generated by STAAD.Pro Physical Modeler (with any necessary load factors).

Note: Do not use primary load cases with STAAD.Pro Physical Modeler reference loads and repeat style load combinations, or the results of those load cases will be included twice.

M. To define a new load combination

To manually define a load combination for post-processing, use the following procedure.

One or more primary load cases or load combinations must be defined in order to create a combination.

Note: Load combinations are not directly processed by the analysis engine. Instead, their results are combined as specified by the combination method. In order to process combinations of load cases use either a Repeat Load or a Reference Load in a new Primary Load case.

1. On the Loading ribbon tab, select the Combinations tool in the Loading group.

   The Define Load Combinations dialog opens.

2. Type the Load No and load combination Name.

3. Select the combination Type to use:
   - Normal - a linear combination of a factor multiplied by a load result value
   - SRSS - a linear combination of either a Normal type or a square root of the sums squared (SRSS) combination. The factors for each control how each load component is added to the resulting combination
   - ABS - a linear combination of a factor multiplied by the absolute value of a load result

4. Select a Primary load case or existing load combination from the Available Load Cases list and then click > to add the load combination definition.

   The selected load case is added with the default type factors defined above. You can change the factors for an individual load combination component as needed.

Moving Loads: In order to include the individual load cases from a moving load generator, you must add these using the STAAD.Pro Editor. They cannot be entered using the dialog as they are not generated until the STAAD engine processes the input file.

5. Repeat step 4 for all load components to be added to this load combination.
Tip: Click >> to include all Available Load Cases, which will be use the default factors.

6. (Optional) To remove a load component from the Load Combination Definition, select the component in and then click <. To remove all load components, click <<.

7. Click Add.

Related Links
• TR.35 Load Combination Specification (on page 2791)
• Define Load Combinations dialog (on page 3003)

M. To define primary load type

To define or change the load type specified for a primary load case, use the following procedure.

You can define or change the load type for any previously added primary load case.

The load type is used for automatic load combinations.

1. On the Loading ribbon tab, select the Primary Load Type tool in the Load Generation group.

2. For each primary load case, select the Type from the drop-down list in the cell.

3. (Optional) For Life or Roof Life load types, check the Reducible-IBC 2003 if this particular load is considered a reducible live load per IBC.

4. Repeat steps 2 and 3 for all primary load cases.

5. Click OK.

Related Links
• Define Load Type dialog (on page 3032)
• TR.32 Loading Specifications (on page 2650)

M. To define automatic load combination rules

To define new or to edit existing rules for generating automatic load combinations, use the following procedure.

Note: In order to properly generate load combinations for Eurocode (unofficially referred to as EC0), the program contains a macro which allows for gamma inputs. For that code, refer to “M. To generate load combinations per Eurocode (on page 870).”

1. On the Loading ribbon tab, select the Automatic Combinations > Edit Auto Combination Rules tool in the Load Generation group.

The Edit Loading Rules for Auto Load Combination Generation dialog opens.

2. (Optional) To create a new Code and Category of load combinations:
a. Click **New Code**. The *Add New Code* dialog opens.

![Add New Code](image)

b. Type a **Code Name** and then click **OK**. A new, empty table is loaded.

c. Click **New Category**. The *Add New Category* dialog opens.

![Add New Category](image)

d. Type the **Category Name** and the **No. of Rules** to initially create in the table. You can add or remove table rows after you create the table.

e. Click **OK**. The specified rows are added for the new table.

3. For each load type (i.e., column):

    a. Set the **Include Notional Loads?** check box to include notional loads in this type for direct analysis. Notional loads are included by default only in dead, live, roof life, snow, and ice load types.

    b. Select the appropriate **Combination Rule** option for this load type:

        • **Combine all cases together** - For each rule, a single combination will be created which will include all the load cases of that load category multiplied by the factor in the table.

        • **Separate combination for each case** - For each rule, multiple combinations will be created, each will include one of the load cases of that load category multiplied by the factor in the table.

        • **All possible combinations** - For each rule, multiple combinations will be created which will include each of the load cases of that load category on their own and with each and every other load case of that category multiplied by the factor in the table.

4. For each combination (i.e., row) in the table, specify the load multiplier for the load types present in that load combination.

**Tip:** To add a new road, click **Add Row**. To remove a row, click the row number to highlight the entire row and then press `<Delete>`.
5. To save the changes to a table, click **Update Table**.
6. Repeat steps 3, 4, and 5 individually as needed to specify the load combinations for the code category.
7. Click **Close**.

You can now generate load combinations using this rule set.

**Related Links**
- [Edit Load Rules for Auto Load Combination Generator dialog](#)
- [Edit Load Rules for Auto Load Combination Generator dialog](#)

M. To automatically generate load combinations

To automatically generate load combinations based on predefined rules for load types, use the following procedure.

In order to use this feature, one or more primary load cases must be created with a load type defined. A load type can be assigned to a primary load case either at the time when the load is being created or later.

If you want to customize the load combination rules or create your own for your organization, you must do this before creating the automatic load combinations.

**Note:** In order to properly generate load combinations for Eurocode (unofficially referred to as EC0), the program contains a macro which allows for gamma inputs. For that code, refer to “M. To generate load combinations per Eurocode” (on page 870).

1. On the **Loading** ribbon tab, select the **Automatic Combinations** > **Automatic Load Combinations** tool in the **Load Generation** group.

   The **Auto Load Combination** dialog opens.

   **Note:** If no load cases have yet been defined for the model, a warning message is displayed over the dialog and the parameters are inactive. Primary load cases are defined using the **Create Primary Load Case** dialog.

2. Choose the code to use from the **Select Load Combination Code** drop-down list.
3. Chose the code-specific load combination table to use from the **Select Load Combination Category** drop-down list.
4. (Optional) Type the **Select Starting Combination No** to use for the first generated load combination.
5. (Optional) Check the **Create Repeat Load Cases** option to generate repeat load cases in place of results load combinations.
   Standard load combinations are algebraic combinations of results and therefore are not appropriate for second order effects. A repeat load case is a new primary load case which will re-use the loads from previously defined primary load cases in the analysis engine.
6. (Optional) Check the **Include Notional Load?** option to include notional loads (when notional loads have been specified) in the Repeat Load Cases.
7. Click **Generate Loads**.
   The load combinations based on the selected code and category are displayed in the **Selected Load Combinations** list.
8. Select any loads you do not want included in the load combinations and then click < to place them in the **Discarded Load Combinations** list.
9. Click Add.

Related Links
• Auto Load Combination dialog (on page 3033)
• TR.35 Load Combination Specification (on page 2791)

M. To generate load combinations per Eurocode

To generate load combinations for the Strength limit state per Eurocode - Basis of structural design, BS EN 1990:2002+A1:2005 (sometimes referred to as “Eurocode 0”).

The load combination generator is capable of creating load combinations per equations 6.10, 6.10a, or 6.10b found in Cl. 6.4.3.2.

These equations specify the following combinations of loads:

\[ \sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_p P + \gamma_{Q,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,1} \psi_{0,1} Q_{k,1} \]

Alternatively, for the strength limit state, the less favorable of equations 6.10a and 6.10b may be used:

\[ \sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_p P + \gamma_{Q,1} \psi_{0,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,1} \psi_{0,1} Q_{k,1} \]

(6.10a)

\[ \sum_{j \geq 1} \xi_{j} \gamma_{G,j} G_{k,j} + \gamma_p P + \gamma_{Q,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,1} \psi_{0,1} Q_{k,1} \]

(6.10b)

where

\[ G_k = \text{Permanent actions} \]

\[ P = \text{Prestress actions} \]

\[ Q_k = \text{Variable actions} \]

Note: The effects in each of the above equations are always additive. If any effect is negative (that is, would reduce the final sum), its effect is taken as zero.

1. On the Utilities ribbon tab, select the User Tools > Euro Code Load Combination Generator tool in the User Tools group.

The Eurocode Combination Generator dialog opens.
2. Select the **Equation Number** to use for the generation of load combinations.

3. (Optional) Specify a **Start Combination Number**.

4. Select the **Method** as either:
   - **Combination** to use post-analysis combination of results, which appropriate for linear elastic analysis
   - **Repeat Load** to generate repeat load cases from the primary load cases, which is appropriate for second order effects and dynamic analysis

5. (Optional) Specify load Factors for use in the combination equations.

<table>
<thead>
<tr>
<th>Factor title</th>
<th>Equation notation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Permanent Actions, Gamma G</td>
<td>( \gamma_G )</td>
</tr>
<tr>
<td>Pre Stress Actions, Gamma P</td>
<td>( \gamma_P )</td>
</tr>
<tr>
<td>Variable Actions, Gamma Q</td>
<td>( \gamma_Q )</td>
</tr>
<tr>
<td>Permanent Action Reduction Factor, ( \bar{X} )</td>
<td>( \xi )</td>
</tr>
</tbody>
</table>
### Factor title | Equation notation
---|---
Variable Action Combination Factor, Psi 0 | $\psi_0$

**Note:** The default values are taken from those provided in EC0.

6. **(Optional)** Click Table A1.1 to select one of the recommended values for the Permanent Action Reduction Factor, Psi ($\psi_0$).

7. Click **Categories** to specify in which load action classification each STAAD.Pro load category is to be assigned.

<table>
<thead>
<tr>
<th>Action</th>
<th>Included Loads</th>
</tr>
</thead>
<tbody>
<tr>
<td>(None)</td>
<td>Loads of this type will not be included in generated load combinations.</td>
</tr>
<tr>
<td>Permanent</td>
<td>Gk (permanent) By default, Dead loads are included.</td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> You may also want to include Mass or Gravity load.</td>
</tr>
<tr>
<td>Variable</td>
<td>Q (variable) By default, Live, Roof Live, Wind, and Snow are included.</td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> You may also want to include loads such as Seismic, Temperature, etc.</td>
</tr>
<tr>
<td>Pre Stress</td>
<td>P (prestress) No loads are included by default in this category.</td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> As STAAD.Pro does not use a load category for prestressing forces, a less-common load category such as Imperfection can be used for this action.</td>
</tr>
</tbody>
</table>

8. Click **Generate Combinations**.
A confirmation dialog box opens with the status of the export. Load cases are generated with the selection equation and load case number in the title.

**M. To add a repeat load case**

To create a primary load case using combinations of previously defined primary load cases, use the following procedure.

A repeat load case is treated as a new primary load. That is, it is processed by the analysis engine rather than being a combination of results. It is therefore applicable for P-Delta analysis.

A repeat load is added similar to a load item within a primary load case.

1. **Either:**

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   **Tip:** This will add the load item to the currently selected load group selected in the program status bar.
   
   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.
The Add New Load Items dialog opens.

2. Select the Repeat Load tab.
   A list of the primary load cases which occur before the current primary load case is displayed.

3. Either:
   - select a load case on the left and then click \( > \) to add it to the repeat load definition
   - or
   - click \( >> \) to add all the primary load cases to the repeat load definition

4. (Optional) Type a load Factor to use for each primary load case included in the repeat load definition.

   **Note:** The primary load case data is factored rather than results.

5. Click Add.

6. Click Close.

**Related Links**
- TR.32.11 Repeat Load Specification (on page 2770)
- Repeat Load tab (on page 3028)

### M. To create a reference load

To define a reference load and add it to a reference load item, use the following procedure.

A reference load case allows you to construct a load case with load items but it will not be directly solved by the analysis engine. It is only solved when it is called by another load case. This allows you to limit the number of load cases analyzed.

**Tip:** A reference load of type Mass is especially useful to define the structure masses used for dynamic analyses. One reference load case can be used by each of those analysis commands.

1. In the Load & Definition dialog, select the Definitions > Reference Load Definitions entry and then click Add.
   The Add New Reference Load Definitions dialog opens.

2. (Optional) Type a Number for the load case.
   The program will increment the load case number based on existing load case numbers.

3. (Optional) Select the Loading Type.
   This selection is used when automatically generating load combinations. If you select a live load type, you may also indicated if the load is Reducible per UBC/IBC.

4. Type a Title used to easily identify this load case.

5. Click Add.
   The reference load definition is added to the Definitions > Reference Load Definitions entry in the Load & Definition dialog.

6. Add load items to the reference load definition.
   This is done similar to how you would add load items to a primary load case. Reference load definitions can contain:
   - selfweight loads
   - nodal loads
   - member loads
7. Either:

On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

**Tip:** This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

8. Select the **Repeat Load > Reference Load** tab.

A list of the reference load definitions is in the **Available Load Cases** list.

9. Either:

select a reference load definition and then click > to add it to the Reference Load item

or

click >> to add all the available reference load definition to the Reference Load item.

10. Repeat Step 9 to add additional reference load definitions to the Reference Load item.

11. (Optional) Type a Factor to apply to each reference load definition in the Reference Load item as needed.

12. Click **Add**.

**Related Links**
- **Repeat Load tab** (on page 3028)
- **TR.31.6 Defining Reference Load Types** (on page 2642)

### M. Damping Modeling

In STAAD.Pro, you can specify damping for the entire structure, composite damping for each material in the structure, or damping for each mode. Modal damping can be explicitly defined for each mode or calculated by the program based on the first two frequencies.

#### M. To assign a composite damping ratio

To assign a damping ratio to a model selection, use the following procedure.

This method assigns a composite damping ratio by a material constant.

**Note:** If you assign a material definition (on page 792) to members, elements, or solids, then the critical damping ratio for that material will be used.

1. On the **Specifications** ribbon tab, select the **Constants > Damping Ratio** tool in the **Materials** group.
The **Material Constant - Damping Ratio** dialog opens.

2. Select either:

   - the **Enter Value** option and then type the composite damping ratio value (must be between 0.001 and 0.99)
   - or

   one of the three predefined materials: **Aluminum**, **Concrete**, or **Steel**

<table>
<thead>
<tr>
<th>Material name</th>
<th>Composite damping ratio value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>0.03</td>
</tr>
<tr>
<td>Concrete</td>
<td>0.05</td>
</tr>
<tr>
<td>Steel</td>
<td>0.03</td>
</tr>
</tbody>
</table>

3. Select the **To Selection** option to limit the assignment to the selection set.

4. Click **OK**.

**Related Links**
- [G.17.3.3.1 Composite Damping](#)
- [Material Constant dialog](#)
- [TR.26.2 Specifying Constants for Members and Elements](#)

**M. To explicitly define damping values for modes**

To define damping values for modes for use with dynamic analysis, use the following procedure.

1. On the **Loading** ribbon tab, select the **Modal Damping** tool in the **Dynamic Specifications** group.

   The **Modal Damping** dialog opens.

2. Select the **Explicit** option.

3. Type a **Damping** ratio to use for each mode number.

4. Click **OK**.

**Related Links**
- [G.17.3.3.2 Modal Damping](#)
- [Modal Damping dialog](#)
- [TR.26.4 Modal Damping Information](#)

**M. To evaluate damping for modes**

To instruct the program to evaluate the modal damping values from the first two...
STAAD.Pro also has a similar method available for calculating the modal damping when provided the mass-proportional damping coefficient, \( \alpha \) and the stiffness-proportional damping coefficient, \( \beta \). This method must be input using the STAAD.Pro Editor.

1. On the **Loading** ribbon tab, select the **Modal Damping** tool in the **Dynamic Specifications** group.

   ![Modal Damping dialog](image)

   The **Modal Damping** dialog opens.

2. Select the **Evaluate** option.

3. Type **Min** and **Max** values to evaluate damping values for each mode.

   The minimum damping value is used for the first two modes. This is also used to evaluate the damping coefficients for subsequent modes based on the first two modal frequencies. If any evaluated damping is greater than the specified maximum value, then the maximum damping is used instead. Refer to **Evaluate Damping** (on page 2370) for additional details.

4. Click **OK**.

**Related Links**
- **G.17.3.3.2 Modal Damping** (on page 2368)
- **Modal Damping dialog** (on page 3059)
- **TR.26.4 Modal Damping Information** (on page 2510)

### M. Mass Modeling

#### M. To add a mass model reference load

A mass model reference load is added in the same way as any other reference load, but with the Loading Type set to Mass.

1. Either:

   - on the **Loading** ribbon tab, select the **Reference Load Case** tool in the **Loading Specifications** group

   ![Reference Load Case tool](image)

   or

   - select the **Definitions > Reference Load Definitions** section of the **Load & Definition** dialog box and then click **Add**.

   ![Add reference load definition](image)

   or

   - (Optional) Type a reference load identification number in the Number field.
Tip: This number is incremented by one from any previously defined reference loads and typically does not need to be changed.

3. Select Mass as the Loading Type.
4. (Optional) Type a label in the Title
   For example, you may want to label the reference by typing Mass Model.
5. Click Add.
   The dialog box closes and a new reference load definition is added to the input file.

Related Links
• TR.31.6 Defining Reference Load Types (on page 2642)

M. To add mass loads to the mass model reference load

1. Select the mass Reference Load Case in the Load & Definition dialog box.
2. Click Add.
   The Add New: Reference Load Items dialog box opens.
3. Select the type of load you want to add from the tree.
4. Specify load parameters (e.g., magnitude, direction, etc.).
5. Click Add.
6. (Optional) Repeat steps 3 through 5 to add additional loads.
7. Click Close.

Related Links
• TR.31.6 Defining Reference Load Types (on page 2642)

M. To create a load envelope

To create a load envelope for grouping loads together in the analytical workflow, use the following procedure.

1. Select the Load Envelopes list on the Load & Definition dialog and then click Add.
   The Add New Load Envelopes dialog opens.
2. (Optional) Select an envelope Type from the drop-down list.

   Note: In the case of designing steel members for strength and serviceability, it is important to select the applicable load type.

3. Either:
   select a load case or load combination in the Available list and then click > to add that item to the envelope
   or
   click >> to add all the Available load cases and load combinations to the envelope
4. Repeat step 3 to add additional load cases or combinations to the envelope.
5. Click Add.
   The load envelope is added to the input file.
6. Repeat steps 2 through 5 to create additional load envelopes.
7. Click Close.

Related Links
The STAAD.Pro Piping workflow is used to import piping support geometry and reactions from Bentley AutoPIPE.

The piping system in a plant model is supported by a number of structural components – such as pipe racks, cradles or shoes – constructed exclusively for that purpose or from the existing beams, columns and slabs of the structure. To facilitate the design of structures to sustain the loading imparted by the piping system, STAAD.Pro includes a seamless transfer of support reactions calculated from an AutoPIPE analysis to the structure modeled in STAAD.Pro.

**M. Using the Piping Workflow**

**General Workflow**

1. Import a piping model via the PipeLink utility.
2. Use the Support Connection Wizard to automatically assign connections between pipe nodes and structural entities.
3. Modify or add connections as necessary either graphically or via the Pipe Support table.
4. Transfer loads from the pipe stress model to the STAAD.Pro model.
5. Analyze the STAAD.Pro model for the new loads.
6. Export any changed or updated model data back to AutoPIPE via the PipeLink utility or the Export Macro.

**M. To import a piping model**

**Tip:** The scale, orientation, and origin of both the piping model and structural model should be the same.

1. From within Bentley AutoPIPE, export the pipe stress model using the PipeLink utility.

   **Note:** Refer to the AutoPIPE documentation for additional assistance on using this product.

2. Open a corresponding model in STAAD.Pro.
3. Select the **Piping** workflow.
   The Piping ribbon tab and pages open.
4. Either:
   - on the **Piping** ribbon tab, select **Interop > Import** in the **Models** group
   - click **Import** on the **Pipe Model dialog** (on page 3111).
   An **Open** dialog opens with the **Files of type** filter set for PipeLink files.
5. Select the **PipeLink file created from AutoPIPE** in the Files list box.

6. Click **Open**.

   A message dialog displays the status of the file import process.

   **Note:** If clashes are detected when importing, then the **Merging Support Connection dialog** (on page 3120) opens. This is used to specify if either the imported or local data should be used.

7. Click **OK** to dismiss the dialog.

   The **Support Connection Wizard** opens.

   **Tip:** Multiple models can be imported to a single STAAD.Pro project. Any actions are performed on the current active model as selected in the **Pipe Model** dialog.

   **Tip:** Imported data is saved with the structure upon exiting the Piping mode for later use.

**Related Links**
- **Pipe Model dialog** (on page 3111)
- **Merging Support Connection dialog** (on page 3120)
- **Support Connection Wizard** (on page 3113)

**M. To use the Support Connection Wizard**

1. Either:

   on the **Piping** ribbon tab, select the **Support Connection Wizard** tool in the **Connection** group

   or

   import a piping model (on page 878) as described above.

   The **Move Pipe Model** dialog opens.

2. Enter the offset distance between pipe and structural models and the click the **Next >** button.

   The **Pipe Nodes** dialog opens.

   The filtering options are All, Connected, Unconnected, V-Stop, or Anchor (relating to whether a connection to the STAAD.Pro model has been defined). When support type information is available the filter will be expanded to include these as well.

3. Select which nodes will be used by adding them to the Selected list and click the **Next >** button.

   The **Structure Beams** dialog opens.

   The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group, View and Property.

4. Select which structure elements will be used by adding them to the Selected list and click the **Next >** button.

   The **Structure Node** dialog opens.

   The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group and View.

5. Select which structure nodes will be used by adding them to the Selected list and click the **Next >** button.
The Parameters dialog opens.

6. Here you will set some parameters so the wizard can establish pipe connections and click the Next button.

The results of the Wizard will be displayed in the final dialog.

See "Parameter Page / Connection Finder" for additional details on these settings.

7. Review these results and if changes are necessary, you may click the Back button. Otherwise, click the Finish button to accept.

The results will be presented in a table with columns for pipe node id and for the structural item it is to be connected to. This second column will provide a drop list allowing the user to choose either “No connection” or one of up to 5 closest items. The items will be listed with the closest at the top. The initial state will be the closest item or ‘Ground’, if no good matches were found.

**Tip:** Manual connections can be generated using the following procedures.

**Related Links**
- Support Connection Wizard (on page 3113)

**M. To draw connections between piping supports and the structure**

1. On the Piping ribbon tab, select the Connection Support tool in the Connection group.

   ![Connection Support Tool](image)

   The mouse pointer changes appearance to the Support Connection cursor ( ).

2. Click a piping model support.

   A dynamic line is now attaches this support to the cursor.

3. Click either a node or a beam element in the structural model.

   The support association is shown as a dashed, red line in the view window. The Pipe Supports table (on page 3117) is updated to reflect the Connect To structural entity.

   **Note:** You may wish to edit the Dist. Along Beam value in the Pipe Supports table to control the exact connection location.

4. Repeat steps 3 and 4 to connect as many supports as needed.

5. Press the <ESC> key to exit the Support Connection cursor.

**Related Links**
- Pipe Supports table (on page 3117)

**M. To manually specify connections between piping supports and the structure**

1. Select the Supports page in the Piping workflow.

   The Pipe Supports table (on page 3117) opens.

2. Select the row of the Pipe Support Node you wish to connect to the structure.

3. Select if the support is Connected To either a Node or Beam.
**Note:** Ground supports are the default and are assumed to *not* be supported by the structural model.

4. Press <Tab>.
5. Type the connecting Beam or Node number in the **Structure Entity** cell for the Pipe Node.
   The connection association is shown as a dashed, red line in the view window.
6. Press <Tab> to update the model.
7. (Optional) (For Beam supports) Specify a distance along the beam, from the start node, to the connection point and then press <Tab>.
8. Repeat Steps 2 through 7 to connect as many supports as needed.

**Related Links**

- *Pipe Supports table* (on page 3117)

---

**M. To transfer load data for structural analysis**

1. On the **Piping** ribbon tab, select the **Transfer Loadings** tool in the **Loading** group.

   ![Transfer Loadings](image)

   The **Transfer Pipe Reactions to Structure Model** dialog (on page 3119) opens.

2. Make any changes to the load data as needed.
3. Click **OK**
   A Transfer Loads summary dialog opens to inform you of the status of the load data transfer.
4. Click the **OK** button to dismiss the dialog.
   The loads from the pipe stress supports are now added to the **Load & Definition** dialog in the model.

   **Note:** The structure is now ready to be analyzed for the imported pipe support loads.

**Related Links**

- **Transfer Pipe Reactions to Structure Model dialog** (on page 3119)

---

**M. To export model data for use in AutoPIPE**

Before you export the model data, you should analyze the structural model. Select the **Analytical Modeling** workflow and then on the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group. Return to the **Piping** workflow once the analysis is sucessfully completed.

1. On the **Piping** ribbon tab, select **Interop > Export** in the **Models** group.

   ![Interop Export](image)

   The **Export Revised Model** dialog (on page 3110) opens.

2. (Optional) Edit **User Name**
3. Click **OK**
   The **PipeLink for STAAD.Pro V8i** utility program opens.
4. Either:
Select an existing data exchanger file (file extension .pipelink) by clicking [...] or Create a new file by clicking Use Blank.

5. Either:
   - Select Export > Start Run.
   - or
   - Select the Run tool.

   The progress of the export process is displayed in the Output window. A message dialog opens to provide you with the status of the export.

6. Click OK.
7. Review any error or warning messages as needed.
8. Select File > Exit to close the utility program.

**Note:** Refer to the documentation included with the PipeLink plug-in for additional help in using this utility.

**Related Links**
- Export Revised Model dialog (on page 3110)

---

**M. To export STAAD.Pro model data into AutoPIPE**

**Tip:** The macro ToAutoPipePub.vbs is included with STAAD.Pro for the purpose of creating structural support data in AutoPIPE from a STAAD.Pro model. The resulting file contains only the support frame data for the current STAAD input file.

1. On the Utilities ribbon tab, select User Tools > Export Model to AutoPIPE in the User Tools group.

   The Export STAAD Model to AutoPIPE dialog (on page 3107) opens.
2. Select the AutoPIPE Neutral file (file extension .NTL) you wish to create or to which you wish to add data.

   **Tip:** Click [...] to select an existing AutoPIPE Neutral file.
3. Specify Job Information and Header data as necessary.
4. Select the appropriate system of Units.
5. Select an option for how the Member Properties are displayed in AutoPIPE.
6. Click OK.

   **Note:** If any invalid structural properties or specifications are found, a warning dialog opens to display these issues. Otherwise, a dialog opens to confirm the AutoPIPE Neutral file has been created.

**Related Links**
- Export STAAD Model to AutoPIPE (on page 3107)
M. Pages in the Piping Workflow

The Pages in the Piping workflow are described below in brief.

Table 46: Piping Page Controls in STAAD.Pro

<table>
<thead>
<tr>
<th>Page</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Piping</td>
<td>Used to display piping models and select the active model when more than one piping model has been imported. When the Piping page is selected, the Pipe Model dialog (on page 3111) opens.</td>
</tr>
<tr>
<td>Supports</td>
<td>Used to review and edit support connections manually. When the Supports page is selected, the Pipe Supports table (on page 3117) opens.</td>
</tr>
</tbody>
</table>

M. Bridge Deck workflow

STAAD.beava (Bridge Engineering Automated Vehicle Application) mode is integrated into the Bridge Deck workflow. This facility is used for generating loads for use in the analysis of bridge structures.

Overview

The general philosophy governing the design of bridges is that, subject to a set of loading rules and constraints, the worst effects due to load application should be established and designed against. The process of load application can be complex as governing rules can impose interdependent parameters such as loaded length on a lane, lane factors, and load intensity. To obtain the maximum design effects, engineers have to try many loading situations on a trial and error basis.

This leads to the generation of many live load application instances (and a large volume of output data) that then must be combined with dead load and other effects, as well. Bridge Deck is used to minimize the load application process while complying with national code requirements.

The program is based on the use of influence surfaces, which are generated by STAAD.Pro as part of the loading process. An influence surface for a given effect on a bridge deck relates its value to movement of a unit load over the point of interest. The influence surface is a three-dimensional form of an influence line for a single member (or, in other words, it is a 2D influence function).

STAAD.Pro will automatically generate influence surfaces for effects such as bending moments for elements, deflection in all the degrees of freedom of nodes, and support reactions. You then instruct the program to utilize the relevant influence surfaces and, with due regards to code requirements, optimize load positions to obtain the maximum desired effects.

Once the influence surfaces have been generated, they are saved and can be reused for any further investigation that may be required. These remain valid as long as the you do not altered the structural model. Changes to the structural model can alter the pattern of the influence surfaces and you must ensure that a further analysis takes place before any further processing.

Your engineering knowledge and judgement is critical in deciding which effects are required and at what position to obtain them. This is where you can save a lot of processing time and also can ensure critical positions are not missed.
**Bridge Standards**

The Bridge Deck mode supports the following standards:

- **UK**: BS 5400 Part 2 (on page 3132)
- **UK**: BD21/01 Chapter 5 (on page 3133) and Annex D (on page 3133)
- **American**: AASHTO ASD/LFD (on page 3134)
- **American**: AASHTO LRFD (on page 3135)
- **Indian**: IRC Chapter 6 (on page 3135)

All the relevant code instructions for loading definitions and traffic lane calculations are incorporated in BEAVA and, in cases where vehicle axle arrangements are not standard, it is possible to define a vehicle and save it in the library for use it in the analysis.

**Note**: It is not uncommon for local authorities to have superseding documents or amendments to national documents. It is the responsibility of the engineer to be familiar with these codes and input the appropriate parameters for loads.

BEAVA is fully integrated in STAAD.Pro and utilizes the same GUI for all input and output data.

**Roadway and Load Modeling**

You must define the width of the Roadway as straight or curved parallel lines. BEAVA then automatically calculates the following in accordance with the selected code:

- Number of Notional Lanes (Traffic Lanes)
- Influence lines along the center line of notional lanes
- Loaded length along the Lanes
- Critical location of uniformly distributed load
- Critical location of knife edge load
- Critical location of vehicle load
- Maximum effect value
- Associates effects values

Once the program has completed calculating the above, a text file containing the results is displayed on the screen. You can also then examine the results graphically.

Loading arrangements for the effects requested can be displayed on the model and, for every loading arrangement produced, you can instruct the program to generate a STAAD.Pro load case. The added live load cases can be combined with dead loads using the normal STAAD.Pro load combination generation. The final model can then be analyzed in STAAD.Pro and then post-processed.

**M. Using BEAVA**

There are a number of distinct stages in the use of the program. It is strongly advised to follow this order when using the program.
Tip: It is recommended that you work through the EX. Bride Deck Loading Example (on page 5033) to familiarize yourself with this process.

M. To open a model in the bridge deck workflow

1. Create a structural model—including member properties and support conditions— in STAAD.Pro.
2. Run an analysis on the structure.
3. Select Bridge Deck in the Workflows panel.

   Note: If you do not have a license for this module, you will not be able to proceed.

   The Bridge Deck ribbon tab opens.

M. To define a bridge deck

1. (Optional) If the mouse pointer is not already displayed as the Plate Cursor, in the Analytical Modeling workflow on the Geometry ribbon tab, select the Plates Cursor tool in the Selection group.

2. Select the elements and/or members that will be used to define the bridge deck area of the models.
3. In the Bridge Deck workflow on the Bridge Deck ribbon tab, select the Create Deck tool in the Deck group.

   The Save Deck as dialog opens.
4. (Optional) Type a Name for the deck.
5. Click OK.

M. To generate influence surfaces

To generate influence surfaces for the currently selected deck, do the following.

1. On the Bridge Deck ribbon tab, select the Loading > Influence Surface Generator tool in the Loading group.

A temporary STAAD input file (filename_deck.std) is sent to the STAAD analysis engine for processing. The STAAD Analysis and Design dialog opens to display the progress. When the analysis is complete, the dialog closes automatically.

M. To define a roadway

To define a roadway (i.e., carriageway in some regions), do the following.
1. Either:
   Select a deck from the View window.
   or
   On the Bridge Deck ribbon tab, select the Select Deck tool in the Deck group and then select one of the defined decks from this list.

2. On the Bridge Deck ribbon tab, select the Define Roadway tool in the Deck group.

   The Roadways dialog (on page 3123) opens.

3. Click New.
   The Define Roadway dialog (on page 3125) opens.

4. Specify the distances from the origin to each curb edge.
   The preview updates to display the roadway on the currently selected deck.

5. Click OK.
   A new roadway is now added to the Roadways list in the Roadways dialog and is outlined in blue on the deck in the active view window.

6. Click Close.

M. To generate loads on the roadway

1. On the Bridge Deck ribbon tab, select the Loading > Run Load Generator tool in the Loading group.
   The Load Generator Parameters dialog (on page 3130) opens.

2. On the General tab, select the appropriate Design Code.
   The <code> tab updates to reflect this selection.

3. Select the appropriate Limit State.

4. Select the <code> tab.

5. Make the code-specific selections for vehicles, road conditions, multiple presence factors, and impact factors.

6. Select the Node Displacements, Support Reactions, Beam End Forces, or Plate Center Stress tab.

   Note: An action must be defined for which the maximum results determine the placement of the moving loads.

7. Specify a structural object number, direction of action, and sign of effect.

8. Repeat steps 6 and 7 as many times as needed.

9. Click OK.
   The program analyzes the deck to obtain the critical load positions for the specified action.

A summary of the analysis is opened in your default text editor (e.g., Notepad). The file (named filename_deckx.out) is saved in the same location as the input file. Close the text editor once you have completed reviewing this file.

The placement of the loads for each effect requested may be reviewed.
M. To transfer the loads to STAAD.Pro

To transfer the generated load cases to STAAD.Pro for analysis and design, do the following.

1. On the Bridge Deck ribbon tab, select the Loading > Create Loading in STAAD Model tool in the Loading group.
   A message dialog opens to confirm the load was successfully added in the STAAD input file.
2. Select the Analytical Modeling workflow.
3. On the Loading page, review the loads added to the input file in the Load & Definitions dialog.

   **Note:** One load was added for each action requested in the Load Generators Parameters dialog.

4. (Optional) Specify additional analysis commands or design parameters as needed.
5. On the Analysis and Design ribbon tab, select the Run Analysis tool in the Analysis group.

M. Checking Your Model

STAAD.Pro comes with a number of tools used to check your structure and model objects.

M. To check for multiple structures

To check for multiple, independently defined set of entities in your model, use the following procedure.

The STAAD.Pro analysis engine can accommodate multiple structures in a single model. However, this check allows you to confirm the number of detected structures to prevent an unintentionally defined model.

1. On the Utilities ribbon tab, select the Structure Tools > Multiple Structures tool in the Geometry Tools group.

   The List of Structures dialog opens.

2. (Optional) Select a structure name in the list.
   The entities detected in this structure are selected in the view window.

M. To check for warped plates

A warped plate is defined as a four-noded plate whose nodes do not lie on the same plane. This tool detects such plates.

You can set the tolerance for warped plates by first setting the Tolerance for Warped Plate Element Detection value on the Options dialog Tolerance tab. Only plates whose angular deviation exceeds this tolerance value will be displayed.

For this check, warping is defined as excessive angular deviation between all vertex normals.

1. Select one or more plates in the model.
2. Select the **Plate Tools > Check for Warped Plates** tool in the **Geometry Tools** group on the **Utilities** ribbon tab.

If any warped plates are detected, the **Detect Warp Plates** dialog opens. A list of all selected plates and their maximum angular deviation is displayed. The global coordinates for a selected plate are displayed below.

![Detect Warp Plates dialog]

3. (Optional) Click **Highlight**
4. (Optional) Click **Delete**

**Tip:** You can undo this action through the **Undo** tool in the main interface.

5. Click **Close** when you have completed reviewing warped plates.

An example of such an element is demonstrated in the following STAAD input.

```
STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 2
ELEMENT INCIDENCES SHELL
21 1 2 3 4
FINISH
```

Nodes 1, 2, and 3 lie in the XY plane at Z = 0, but node 4 has a Z coordinate of 2, thereby causing the plate to become non-planar.

The warped plate check sub-divides the quad element into two sets of triangles: one with a diagonal from point 1 to point 3 and another diagonal from point 2 to point 4. These triangles do not lie in the same plane so an angle is formed between them at their shared edge. The largest angle between adjacent triangles in the warped quad element is compared against the tolerance threshold. If this tolerance angle is exceeded, then the plate is reported as excessively warped.
M. To check for and remove duplicate entities

To detect the presence of and remove instances of nodes, beams, plates, etc. which are duplicates, use the following procedure.

One definition of duplicate nodes is that they are 2 or more nodes, having distinct node numbers, but the same X, Y, Z coordinates. For example, if node number 5 has coordinates of (7, 10, 0), and node 83 also has coordinates of (7, 10, 0), node 5 and 83 are considered duplicate. However, STAAD allows you to expand the definition of this term to include nodes separated by any distance. When this menu is selected, the following message box pops up, asking the user to specify a tolerance.

Duplicate members are 2 or more members, having distinct member numbers, but connected between the same 2 nodes. For example, if there are 3 members, say, 17, 46 and 75, all connected between the nodes 105 and 117, those 3 members are considered duplicate. The order of the node incidence is disregarded during this check. For example, if member 17 has 105 as its first node, 117 as its second, and member 46 has 117 as its first node, 105 as its second, they are still treated as duplicate.
A similar definition is applicable for duplicate triangular or quadrilateral plates. Plates are considered duplicates if they share the same vertex nodes, regardless of order.

1. Depending on the entity type you are checking, select one of the following tools from the Utilities ribbon tab in the Geometry Tools group:

   - **Nodes**
     - Node Tools > Check Duplicate Nodes
   - **Beams**
     - Beam Tools > Check Duplicate Beams
   - **Plates**
     - Plate Tools > Check Duplicate Plates

   (Nodes only) The **Remove Duplicate Nodes** dialog opens.

2. (Nodes only) Type a distance in the **Enter Tolerance** field and click the **OK**.

   The tolerance is the value used to define the distance at which nodes are to be considered separate or a duplicate. In some cases, the process of creating the model can result in nodes that are a minuscule distance apart. Specifying a tolerance greater than zero allows the program to consider those nodes as duplicates and allows you to process them as needed.

   The program reviews the model for any duplicates of the selected model element type (using the tolerance for Nodes).

   - If no duplicates are found, a message is displayed.
   - If duplicates are detected, the List of Duplicate Nodes / Members / Plates dialog opens. Each entry in the list contains two or more entities that the program has detected to be duplicates.

3. Select a duplicate set in the list and then click **Merge >>** to remove the duplicate from the model.

4. Select one of the items detected as a duplicate in the **Node / Member / Plate To Keep** drop-down list and then click **OK**.

   The remaining duplicates are removed and other model entities which refer to those duplicates now reference the selected (merged) entity.

5. Repeat steps 3 and 4 as necessary to review all duplicates of the entity type selected.

6. Click **Close**.

**M. To detect and remove zero length members**

A zero length member is one of the undesirable consequences of having duplicate nodes in the structure. A member connected between duplicate nodes, which have the same (X,Y,Z) coordinates, will have a length of zero (or nearly zero).

1. On the Utilities ribbon tab, select the Beam Tools > Check Zero Length Members tool in the Geometry Tools group.
   
   The **Zero length beams** dialog opens.

2. (Optional) Type a different tolerance in the **Enter Tolerance** field.

   Since duplicate nodes can be two or more nodes separated by any user-defined distance, zero length members too can be defined as members of any user-defined length. This distance is the tolerance used in the process of detection.

3. Click **OK**.

   If no members of zero length are detected a message is displayed.
If one or more zero length members are detected, the Detect Zero Length Member dialog opens.

4. Select a zero length member in the list and then click Delete to remove the beam.

**Tip:** You can click Highlight to select the member in the View window (though most zero length members are difficult to see unless at a very high magnification). Then select the Properties tool on the Beam ribbon tab to open the Beam dialog for the zero length member.

5. Repeat Step four as necessary until all zero length members are removed.
6. Click Close.

### M. To check for overlapping collinear members

1. On the Utilities ribbon tab, select the Beam Tools > Check Overlapping Colinear Members tool in the Geometry Tools group.

   If no overlapping collinear members are detected, then a message dialog opens with that message.

   If one or more overlapping collinear members are detected, the Overlapping Collinear Beams dialog opens.

2. Select the overlapping members pair and the Highlight option to select the beam(s) in the Active View window.

3. The members in the Active View window can be manipulated or deleted as necessary while the Overlapping Collinear Beams dialog is open.

```
An example of 2 members which would qualify as overlapping collinear are:

STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 13 10 0; 6 -4 10 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
101 5 6
FINISH

Here, members 2 and 101 are overlapping collinear. Member 2 is entirely confined within the span of member 101, and collinear, but they are not attached to each other.

Another example is:

STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 13 10 0; 6 -4 10 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
101 2 5
FINISH

Here, again, members 2 and 101 are overlapping collinear. But even though they are connected to each other at node 2, again member 2 is entirely confined within the span of member 101, and collinear.
```
M. To change a beam incidence

To change the incidences of a selected set of beams, use the following procedure.

1. Select one or more members in the view window.
2. On the Utilities ribbon tab, select the Beam Tools > Beam Incidence tool in the Geometry Tools group.

The Redefine Incidence dialog opens.
3. Select an incidence definition to use:
   - Switch incidences of selected beams
   - Set the incidence order of the selected beams so that the start node is close to the origin than end node
   - Set the incidence order of the selected beams so that the start node is farther from the origin than end node
4. Click OK.

Related Links
- M. Member Orientation (on page 794)

M. To detect and remove orphan nodes

To detect and then remove nodes which are not connected to any other model entity (i.e., “orphan nodes”), use the following procedure.

In the process of modeling, disconnected nodes can be created in your model.

1. On the Utilities ribbon tab, select the Node Tools > Orphan Nodes tool in the Geometry Tools group. Any detected orphan nodes are selected in the view window.
2. To remove these orphan nodes, on the Utilities ribbon tab, select the Node Tools > Remove Orphan Nodes tool in the Geometry Tools group. The detected orphan nodes are deleted.

M. To display the distance between two nodes

To temporarily display the dimension between two nodes in the active View window, use the following procedure.

1. On the Utilities ribbon tab, select the Node Tools > Node to Node Distance tool in the Geometry Tools group.
The mouse pointer changes to the Node to Node Distance cursor.

2. Click on any two nodes in succession in the active View window.
   A dimension call out is added between these two nodes.

3. Repeat step 2 to dimension as many distances as needed.

4. Either:
   
   Press the <Esc> key

   or

   Select the **Node to Node Distance** tool again
   The mouse pointer returns to the normal selection mode.

M. To display beam lengths

To display the lengths of each analytical member, use the following procedure.

To dimension only the members in a selection, select those members before starting this procedure.

1. On the **Utilities** ribbon tab, select the **Beam Tools > Dimension Beams** tool in the **Geometry Tools** group.

   The **Display/Remove Dimensions** dialog opens.

2. Select the option for which beams to be dimensioned:

   **To...**                                           **Select...**
   dimension all the beams in the current view window  Dimension to View
   dimension only the beams you selected before step 1  Dimension to Selected Beams
   dimension a comma separated list or range of beams   Dimension to List

3. Click **Display**.

4. (Optional) To remove the dimensions, select the **Remove** option and then click **Remove**.

5. Click **Close**.

Any new members will not be dimensioned. You must repeat this procedure to display their dimensions.

**Related Links**

- **Display/Remove Dimensions dialog** (on page 3097)

M. To check for negative volume solids

To check for solids that would result in a negative volume in the analysis, do the following.

This check uses the Jacobian matrix determinate to determine if any of the solids have a negative or indeterminate volume, which would result in an analysis error.

1. On the **Utilities** ribbon tab, select the **Solid Tools > Negative Volume** tool in the **Geometry Tools** group.
The List of Solids dialog opens.

2. Select any solid number in the list to select it in the view window.
3. Click Close.

Related Links

- G.5.2 Solid Elements (on page 2319)

**M. Physical Modeling workflow**

You can use STAAD.Pro Physical Modeler to model a “physical” structure and then re-import that data back into the STAAD.Pro environment for loading, analysis, and design.

**M. Using the Physical Modeler**

You can model structure geometry, specifications, and many loads with the STAAD.Pro Physical Modeler.

You must be a physical model with an empty STAAD project. Existing analytical models cannot be opened in the physical modeler.

When you open a model that was created using the physical modeler, you will be prompted if you want to open the model in the physical modeler environment.

1. Select the **Physical Modeling** workflow.

The STAAD.Pro Physical Modeler window opens.

Refer to the **STAAD.Pro Physical Modeler help** for details on how to use that module to model your structure.

**M. To drop the associated physical model**

To remove the associated physical model from the current STAAD input file, use the following procedure.

**Caution:** This procedure cannot be undone. Once a physical model has been dropped, that model can no longer be opened within the **Physical Modeling** workflow.

When a physical model is associated with a STAAD input file, many tools in the **Analytical Modeling** workflow as well as portions of the input file in the STAAD.Pro Editor are disabled. This is intended to prevent making changes which would potentially corrupt the association between the physical and analytical models. However, there are times when you may need to drop this association to make changes to the analytical model.

1. Select the **Analytical Modeling** workflow in STAAD.Pro.
2. On the **Utilities** ribbon tab, select the **Drop Physical Model** tool in the **Physical Model** group.
A warning dialog opens asking you to confirm you want to break this link.

3. Click Yes.

The input file is re-opened in STAAD.Pro as an analytical model only.

M. Building Planner workflow

The Building Planner workflow integrates STAAD Building Planner into STAAD.Pro. This is used to generate concrete building models which can then be analyzed in STAAD.Pro and detailed using the Advanced Concrete Design workflow.

M. To start a STAAD Model in the Building Planner workflow

   The New page opens to the Model Info tab.
2. Specify the File Name, Location, and Units as necessary.
3. Select the Building option for the Type of model.
4. (Optional) Select the Job Info tab to add project member names, dates, project description data, etc.
   You can also associate your STAAD project with a ProjectWise Project here.
5. Click Create.

The STAAD.Pro window closes and the STAAD Building Planner application opens. The Start dialog opens.

6. Enter the project details, including No of Levels and Founding Depth.
7. Click Create Project.
   The New Plan dialog opens.

You are now ready to start adding details for the first floor plan and modeling your building structure.

Related Links
• GS. Start Page (on page 50)

M. Plans

Plans are building floor layouts (slabs, columns, and beams) which can be used for one or more floors in your building model. This allows you to rapidly model a building in which multiple floors have the same geometry, structure, and loads.

M. To Create a New Building Plan

   The New Plan dialog opens.
2. Specify the general details:
   a. Type a **Plan** name.
   b. (Optional) Select a **Floor type**
   c. Type the **Height of Level Above** (floor-to-floor height).
   d. Click the **Assign Levels** drop-down to assign plan names to each level of the building.

   **Note:** The number of levels is specified in the **Start** dialog.

3. Select the **Create Plan Graphically (Do not import)** option to create a new plan.
   Optionally, you may import the plan from a drawing (on page 897), a PlanWin file, or from another plan in this project (if there is another).

4. Specify the plan details and source (i.e., CAD file, drawing an plan in STAAD, or PlanWin plan).

5. Specify the loads on this plan:
   a. Enter the Slab Loading Parameters.
   b. Enter the Beam Loading Parameters.

6. Select the **Concrete Grade** and **Steel Grade** used in this plan.

7. Click **Create Plan**.
   If the plan was not assigned to any levels in Step 2d, then a warning message displays. Click **Yes** to proceed. Otherwise, the **Slab Details (Rectangle)** dialog opens.
8. Specify the slab details:
   a. Type coordinates for the Position
      
      **Tip:** Leave default to accept origin.
   b. Type material, geometry, and loading parameters.
   c. Specify the direction the slab spans: Two Way, One Way, or Distribute on Selected Edges.

   **Tip:** The Length and Breadth values specified determine the recommended default direction.

9. Click **OK**.
   The slab is drawing in the graphical view.

**M. To Import a Building Plan**

The Building Planner can import plan data from other sources, such as PlanWin files or CAD drawings.

**Note:** The plan data must be in the correct format in a CAD drawing (.dxf file format). The correct units and scale must used in the drawing.
Modeling
M. Building Planner workflow

- Slab data must be in a layer named “Slab”. Only closed polylines or light weight polylines can be imported.
- Column data must be in a layer named “Column”. Use circles of any appropriate radius to mark the columns.
- Beam data must be in a layer named “Beam”. Use lines to mark the beam centerlines.

   If you have not yet specified project details, the Start dialog opens. Otherwise, the New Plan dialog opens.

   **Note:** Alternately, you can select Edit > Import Plan File to replace the current plan with imported plan data using the Import Plan File dialog.

2. Specify the general details:
   a. Type a Plan name.
   b. (Optional) Select a Floor type
   c. Type the Height of Level Above (floor-to-floor height).
   d. Click the Assign Levels drop-down to assign plan names to each level of the building.

   **Note:** The number of levels is specified in the Start dialog.

3. Select either:
   CAD Center line Input (DXF)
   or
   Plan from PlanWin file
   or
   Existing plan from current project

4. Navigate to and select the file you want to use for either CAD or PlanWin file options and click Open.
5. If the selected PlanWin file has multiple plans, select the plan name you want to import.
6. Click OK.

M. To Edit Plan Properties

1. Select View > Plan Properties.
   The Plan Properties dialog opens.
2. Type a new Plan Name or Height of Floor Above value.
3. Click OK.

M. Slabs

Parametric slabs are used to model portions of physical slabs in a building floor. This allows you to model thickened slabs or other complex layout geometry easily.

M. To Add a Rectangular Slab

To add a rectangular slab to the current plan, use the following steps.

1. In a plan workspace, select Slab > Create Slab Rectangle.
2. Click a start and end corner for the slab in the graphics window.
   The start corner “snaps” to the nearest existing slab corner. You can input any corner using the X and Y offsets in t.
   A gray rectangle connects the initial point to your mouse pointer to indicate the general direction of the slab. Exact slab dimensions are specified in the next steps, though.
   The Slab Details (Rectangle) dialog opens.
3. Specify the slab details:
   a. (Optional) Type X Offset Y Offset
      Tip: You may use negative numbers here.
   b. Type material, geometry, and loading parameters.
   c. Specify the direction the slab spans: Two Way, One Way, or Distribute on Selected Edges.
      Tip: The Length and Breadth values specified determine the recommended default direction.

   Tip: Cantilevered slabs can be specified using the Distribute on Selected Edges option and clicking a single, supporting edge.
4. Click OK.
The slab is drawn in the Plan | Slab page graphical view.

**Tip:** If the slab has been drawn beyond the extents of the window, right-click anywhere *not* on a slab and select **Zoom Extend** from the pop-up menu.

Describe the results of the action.

5. Repeat Steps 2 through 4 as necessary to continue drawing slabs.

**Tip:** Press `<Esc>` to quit using this tool.

---

**M. To Add an Irregular Shape Slab**

To add an irregularly shaped slab to the current plan, use the following steps.

1. In a plan workspace, select **Slab > Create Irregular.**
2. Click an initial corner for the slab.
3. Click another corner in either a clockwise or counterclockwise progression from the previous corner.
   The **Point Select** dialog opens.

   **Tip:** The **Point Selection** dialog will not open if the program detects you have indicated an existing slab corner.

4. (Optional) Specify an X Offset and Y Offset value (relative to the point highlighted in a red circle), Distance, and Angle value to place the point at another location.
5. Click **OK.**
6. Repeat steps 3 through 5 to select each additional vertex point on the slab.
7. Right-click to stop adding corners.
   The **Slab Details (Irregular)** dialog opens.
8. Specify the slab details:
   a. Type label, material, geometry, and load parameters.
   b. Specify the direction the slab spans: **Two Way**, **One Way**, or **Distribute on Selected Edges**.

   **Tip**: The resulting dimensions determine the recommended default direction. If one or more slab spanning directions is determined to be unsuitable based on the dimensions, it is disabled.

   **Tip**: Cantilevered slabs can be specified using the **Distribute on Selected Edges** option and clicking a single, supporting edge.

9. Click **OK**.
   The slab is drawn in the Plan | Slab page graphical view.

10. Repeat Steps 2 through 9 to continue drawing irregularly shaped slabs.

   **Tip**: Press `<Esc>` to quit using this tool.

M. To Edit Slab Properties

   To edit the parameters of an existing slab, use the following steps.
**Note:** The label or material and load parameters of a slab may be edited. To edit they geometry, you must deleted and redraw a slab.

1. In a plan workspace, select **Slab > Select/Unselect**.
   
   **Tip:** You can use some of the other selection tools to refine your selection of slabs as necessary.

2. Select a slab in the graphical view.
   
   **Tip:** Hold <Ctrl> to select multiple slabs.

3. Either:
   
   Select **Slab > Set Property and Loads for Selected Slab/s**
   
   or
   
   Right-click and select **Set Property and Loads for Selected Slab/s** from the pop-up menu.

   The **Set Slab Property and Loading** dialog opens.

   ![Set Slab Property And Loading](image)

   4. Check the options for each parameter you want to edit for all selected slabs and then type or select the value.

   5. Click **OK**.

**M. Columns**

**M. To Add a Column**

To add a single column, use the following steps.

1. In a plan workspace, select **Column > Create Column**.

2. Click the nearest slab corner or edge where you want to place a column.

   The nearest slab corner is highlighted with a red circle. The **Column Details** dialog opens.
3. Type column details:
   a. (Optional) Type a custom **Mark**
      
      **Note:** This must be unique to the plan.
   
   b. Type a face **Size** (square column).
      
      All initial column sizes are assumed to be square. Columns can later be resized manually or designed.
   
   c. (Optional) Type an **X Offset Y Offset**
      
      The coordinates of the column are displayed for reference.
   
   d. (Optional) Type an **Angle Distance**

4. Click **OK**.
   
   The column is drawn in dark blue on the graphical view.

5. Repeat Steps 2 through 4 to continue placing columns.

### M. To Add Columns Automatically to the Entire Plan

To place columns automatically at detected points on the entire plan, use the following steps.

1. In a plan workspace, either:
   
   select Column > **Auto Column**, or
   
   or
   
   select Column > **Locate Column**.

   Columns are added at all slab corners on the floor, unless you selected the Locate Column option. In that case, the first detected column location is selected and the Locate Column dialog opens.
2. (Optional) If you selected the Locate Column feature, either:
   Specify the current column mark, offset, and orientation, or
   or
   click **Skip** to skip to the next detected column location, or
   or
   click **Skip Line** to skip all the detected locations in the current column line, or
   or
   click **End** to stop adding columns.

3. Repeat Step 2 as needed until you have reached the end of the detected column locations.

If you used the auto column feature or if you neglected to skip a location, you may need to delete any unwanted columns (e.g., those added at the end of a slab intended to cantilever), move columns, or edit the parameters of columns which are different than the default values for the plan.

M. Beams

M. To Create a Beam

To add a single beam, use the following steps.

**Note:** The **Auto Beam** tool will delete any existing beams on the plan. You may want to use the Auto Beam feature prior to drawing individual beams.

1. In a plan workspace, select **Beam > Create Beam**.
2. Select the start and end support points for the beam.
If beams are drawing over columns or existing beams, the new beam will be broken into separate objects once placed.
The nearest slab corners are highlighted with red circles. The **Beam Details** dialog opens.

3. Type the beam details:
   a. (Optional) Type a custom **Mark**
      
      **Note:** This must be unique to the plan. If you are drawing multiple beams over columns or existing beams, the mark will be incremented for the additionally created beams.
   
   b. Type a **Breadth** and **Depth** for the beam size.
   c. Select the **Grade** of material.
   d. (Optional) To include the uniform dead load for the beam, type a value or click **Compute Beam Load**
      
      **Note:** If Compute is used, the Beam Load dialog opens with the specified beam dimensions. You can specify a material Density and include options for walls supported by the beam.
4. Click **OK**.
   The beam is drawn in green on the graphical view.

5. Repeat Steps 2 through 4 to continue placing beams.

### M. To Add Beams Automatically to an Entire Plan

To place beams automatically at detected points on the entire plan, use the following steps.

1. In a plan workspace, select **Beam > Auto Beam**.
   If there are any existing beams, a warning message that they will be deleted opens.

2. Click **OK**.
   The **Auto Beam Details** dialog opens.
3. Type the beam details:

**Note:** The internal and external (exterior slab edge) beams can be specified separately.

a. Type a **Breadth** and **Depth** for the beam sizes.

b. (Optional) To include the uniform dead load for the beam, type a value or click **Compute Beam Load**

**Note:** If Compute is used, the Beam Load dialog opens with the specified beam dimensions. You can specify a material Density and include options for walls supported by the beam.
c. Select the **Grade** of material.

d. Type the **Cover to Rebars** and **Flange Dimensions** used for design.

4. Click **OK**.

Beams are added at all slab edges on the floor.

---

**M. To Specify Beam Continuity**

To specify continuous beams over supports, use the following steps.

Building Planner uses a window to specify continuous beams for the entire plan. It is typically best practice to draw all beams for the plan before performing this procedure.

1. Select **Beam > Modify Beam Continuity**.
   The **Continuous Beam** window opens.

2. You can specify continuous beams individual for finer control or allow the program to autodetect all continuous beams:
To... | Select...
---|---
select individual beam segments to create continuous beams manually | Modify Data > Mark/Unmark
allow the program to detect all continuous beams automatically | Modify Data > Auto Mark Continuous Beam

If you used the auto mark option, then all beams that the program detects as having possible continuity over supports are marked in light purple. Proceed to Step 5.

3. Click a series of beams you want to mark as continuous. Each beam selected is highlighted in light red. You can click a beam segment again to unmark it.
4. Select Modify Data > Create. The selected beam segments are marked in light purple.
5. Repeat Steps 2 through 4 as necessary to continue to mark individual beam lines as continuous.
6. When you are finished marking continuous beams, either:
   - click the X in the top, right-corner of the Continuous Beam window, or
   - press <Alt+F4>

If you have marked beams as continuous, you must finalize the plan including continuity.

### M. To Edit Beam Properties

To change beam properties, use the following steps.

**Tip:** If the Beam Loading Errors window indicates that beams intended to be cantilevered are not supported, simply right-click on the beam and select Cantilever from the pop-up menu.

   **Tip:** You can use some of the other selection tools to refine your selection of beams as necessary.
2. Select a beam in the graphical view.
   **Tip:** Hold <Ctrl> to select multiple beams.
3. Either:
   - select Beam > Set Property and Loads for Selected Beam/s, or
   - right-click and select Set Property and Loads for Selected Beam/s from the pop-up menu

The Set Beam Property and Loading dialog opens.
4. Check the options for each parameter you want to edit for all selected beams and then type or select the value.

5. Click OK.

M. To Specify Beam Design Parameters

To control some of the various design parameters, use the following procedure.

1. In a plan workspace, select Analysis > Design Parameters.

   The Design Parameters dialog opens.

2. Make changes to the design parameters as necessary.
3. Click OK.

**M. Frames**

**M. To Finalize a Plan**

To finalize a plan for use in a building frame, use the following steps.

In order to complete a 3D frame model using the plan data, you must first finalize the layout of a plan. If you select the Frame page without having finalized plans, a warning message is displayed.

1. In the plan workspace, either:
   - select **Analysis > Finalize Plan (with Continuity)**, or
   - select **Analysis > Finalize Plan (without Continuity)** to ignore continuity between beams, or
   - A message dialog opens to indicate the plan was finalized. If any errors are detected, they will be highlighted in a pop-up window.

If any changes are made to the plan, you must repeat this procedure.

**M. To Create a 3D Frame**

In order to create or edit 3D frames, any changes to plans must first be finalized.

1. In the 3D Frame workspace, select **Assign > Level Properties**.
   - The **Level Information** dialog opens.
2. Click Add Level to add another level and then click OK to confirm.
3. For each level, click in the Plan column to select a plan name from the drop-down list.

   **Tip:** Use the autofill arrows below the table to copy the plan name to rows above or below a selected row, or to all rows above or below.

4. Click OK. The 3D Frame is created.
5. (Optional) Select View > 3D View.

**M. To Create a Shear Wall**

Shear walls created by the Building Planner are analytically columns with the dimensions of the wall they represent.

1. Select Assign > Define Shear Wall.
2. Click the columns or slab corners to define the start and end of the shear wall in plan.

   The points are highlighted with gray-filled, red circles in the view window. The Define Shear Wall dialog opens.
3. Specify the shear wall parameters: as necessary
   a. Type a Mark to label the wall.
   b. If a shear wall is not continuous from base to roof, clear the check mark in the Create column for that level.
   c. Type an X Offset and Y Offset value to move the wall with respect to the selected start point.
   d. Type B and D dimensions to specify a wall size.

4. Click **OK**.
   The shear wall is drawn in dark red in the plan.

5. Repeat Steps 2 through 4 as necessary to continue adding shear walls.

   **Tip:** Press `<Esc>` to quit using this tool.

## M. To Change Supports

To change boundary conditions for the 3D frame, use the following steps.

The supports are at each column or wall base at the first (lowest) level in the building frame. Initially, all supports are assumed as Fixed.

1. Select **Assign > Support Specification**
The **Support Specification** dialog opens.

2. Select the one or more columns or walls.

   **Tip:** Use the Selection drop-down list to change the selection method.

3. Right-click on a column or wall and select **Support Condition - mark** from the pop-up menu.

   The **Support Information** dialog opens.
4. Select the **Support Options** to use.
5. For **Fixed But** option, either:
   - specify the **Release-Spec**
   - or
   - **Spring-Spec** and spring constant for each degree of freedom as necessary.
6. Click **OK**.
7. Repeat Steps 3 through 6 for each support you want to change.
   - **Tip:** The support type is color-coded in the **Support Specification** dialog.
8. Click the **X** in the top, right-hand corner of the **Support Specification** dialog when you are finished making changes to the supports.
   - The **Support Details** table updates with the changes.

**M. To Specify Member Releases**

To change member end release details for the 3D frame, use the following steps.

All beams are initially assumed as fully fixed at both ends.

1. Select **Assign > Member Release (Beams)**.
   - The **Member Release** dialog opens.
2. (Optional) Select the level from the drop-down list of level numbers in the tool bar.

**Tip:** The current level and plan name are displayed in the status bar.

3. Right-click on a beam and select **Release Details-mark** from the pop-up menu.

**Tip:** The **Member Release** dialog contains some tools to select and make release changes to multiple members, such as all primary or secondary beams or all beams along a column line. These tools allow you to select the levels to which you want to apply these release specifications as well.

The **Beam Release Information** dialog opens.
4. Check the boxes for each degree of freedom to release at either end of the member and type a percent release value for each release direction.

Tip: The default release of 99% is intended to represent a full member release.

5. Click OK.

6. Repeat Steps 2 through 6 for each member you want to edit release specifications.

Tip: The member release specifications are color-coded in the Member Release dialog and indicated with dots on each member.

7. Click the X in the top, right-hand corner of the Member Release dialog when you are finished making changes to the member releases.

The Beam Release Details table updates with the changes.

M. To Change Column Size, Orientation, and Alignment

To change a column size and orientation, use the following steps.

1. Select Assign > Column Sizing and Orientation.

The Column Sizing dialog opens.
2. Select a column in the main graphical view. The selected column is highlighted in a red circle and the Mark is displayed. The column orientation and offset are displayed in the right-hand views.

3. To change the column orientation (i.e., Beta angle):
   a. Click a different axis of orientation in the lower, right-hand graphic or click +90 to rotate the column 90°
   b. Click Update Offset Data & Beta Angle.

4. To change the column offset:
   a. Click one of the column faces in the middle, right-hand graphic.
   b. Type an offset value in the indicated units.
   c. Repeat steps 4a and 4b to specify offsets in a different axis.
   d. Click Update Offset Data & Beta Angle.

5. To change a column size:
   a. Click Re-Size Column.

   The Design Column: <mark> dialog opens.
b. Type initial design parameters in the fields.

**Tip:** You can use the Check Column feature to perform a design check on the current column size.

c. Select a Column Shape.

d. Click Design Column.

The column size (Breadth and Depth) at each floor are updated based on the design parameters and specified building loads.

e. (Optional) You can manually type changes to the table values to further refine the design.

**Tip:** Use the automatic fill buttons below the table to copy values up or down the currently selected row.

f. Click OK.

The column diagrams are updated in the Column Sizing dialog.

6. Repeat Steps 2 through 5 for each column you want to change.

7. Click the X in the top, right-hand corner of the Column Sizing dialog when you are finished making changes to the column sizes and orientation.

The Beam Release Details table updates with the changes.
M. To Modify Wind Parameters

To specify parameters for lateral wind loads, use the following steps. The wind load parameters are input per the following codes:

- UBC 97
- IS 875-1987
- ASCE7

1. **Select Assign > Wind Parameters.**
   
   **Tip:** You can also double-click any of the wind load parameters in the Lateral Load dialog.

   The Wind Parameters dialog opens.

   ![Wind Parameters Dialog]

2. Select the **Design Code** from the drop-down list.

3. Either:
   
   Select the appropriate **City** in the drop-down list (recommended)

   or
Type a Wind Speed value directly.

4. Either:
   Select the Life Span (yrs) for which the building is to be designed from the drop-down list (recommended)
or
   Type a Risk Coeff (K1) value directly.

5. Type Height Below Ground and Parapet Wall values, in the units indicated.

6. Type a Gust Factor value.

7. Select the Terrain Category from the description list.

8. Click OK.

You can now review the calculated wind loads by selecting Frame > Wind Effect. In the Wind Effect dialog, select the different directions to see the distributed wind intensity on each floor. Selecting a column in the table displays the wind intensity values at the top and bottom of each floor along that column.

M. To Modify Seismic Parameters

To specify parameters for lateral seismic loads, use the following steps.

The seismic load parameters are input per the following codes:

- UBC 97
- IS 1893-2002

1. Select Assign > Seismic Parameters.

   The Seismic Parameters dialog opens.
2. Either:

Select the appropriate City in the drop-down list (recommended)

or

Type Seismic Coefficient and Seismic Zone values directly.

3. Specify a Response Reduction Factor for the building:
   a. Click [...] adjacent to the Response Reduction Factor field.
      The Response Reduction Factor dialog opens.
   b. Select the appropriate row in the Building Frame Systems description list.
   c. Click OK.
      The Response Reduction Factor is updated.

4. (Optional) Specify or select additional seismic parameters as necessary:
   a. Type an Importance Factor value.
   b. Type a Damping Factor value.
   c. Select the type of soil on site from the Rock/Soil Factor drop-down list.
   d. Select the Combination Method from the drop-down list.
   e. Type the Base Shear Scale Factor value.

5. Select if brick infill panels are present in the building structure to determine the Natural Time Period.

   Tip: Click Details to see the values used in evaluating Px and Pz.

6. Click OK.
M. To Modify the Load Combinations

The Building Planner mode uses building combinations per the Indian code by default. You can modify, remove, or add individual load combinations as necessary.

1. Select Tools > Load Combination. The Loading Combination dialog opens.
2. Make any changes to the load combinations necessary:
   - To...
     - change the load factor on a load type for a particular load combination
       Do the following...
       type the new load factor in the corresponding cell
     - add a new load combination
       click Add and type the load factors for each load type
     - remove an existing load combination
       select the load combination row and click Remove
     - set the current load combinations as the default in Building Planner mode
       click Set as Default
     - restore the default load combinations (i.e., the code load combinations)
       click Load Default Combinations
3. Click Update.
4. Click [X] to close the Loading Combination dialog.

M. Analysis and Design

M. To Generate a STAAD.Pro Model

To generate a STAAD.Pro input file from your Building Planner workflow physical model, use the following procedure.

The physical model created using the Building Planner workflow can export an equivalent STAAD.Pro input file for use with the analysis and design capabilities in STAAD.Pro.

Tip: You must perform an analysis using STAAD.Pro in order to perform design in the Building Planner or Advanced Concrete Design workflows.

1. In the 3D Frame workspace, select Analysis > Generate Analysis File.
2. Click OK.

   The Space Frame File Generation dialog opens.
Caution: Do not close the dialog by clicking X as this will result in an invalid STAAD input file.

3. Specify the parameters to use for the STAAD input file:
   a. Select the Settings to use for diaphragms, beams, and columns.
   b. Select the load cases to use, including if the seismic load should be (static) Seismic Coefficient Method or Response Spectrum Method.
   c. Select the Building Type.
      For Irregular buildings, both positive and negative direction load cases are generated.
   d. Select whether to Generate EQ Load Using by Joint Weight or Reference Load.

4. Click Generate.
   A warning message opens to inform you that the existing STAAD.Pro model will be overwritten.

5. Click Yes.
   The STAAD.Pro input file is created and opens in the Analytical Modeling workflow.

You can now make changes to the STAAD.Pro file as necessary. You can perform an analysis by selecting the Run Analysis tool on the Analysis ribbon tab or by pressing <Ctrl+F5>.

Related Links
• Space Frame File Generation dialog (on page 925)
**Space Frame File Generation** dialog

Used to specify analytical model settings, building details, and load cases to use for a STAAD input file (file extension .std) generated from the 3D building frame in Building Planner.

### Dialog Controls

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Settings group</td>
<td></td>
</tr>
<tr>
<td>Rigid Diaphragm (Master - Slave)</td>
<td>Applies 'Rigid Diaphragm' in the model.</td>
</tr>
<tr>
<td>Member Offset</td>
<td>Used to consider Member offset in the model.</td>
</tr>
<tr>
<td>Release Torsion for Columns</td>
<td>Releases columns for torsion.</td>
</tr>
<tr>
<td>Release Torsion for Beams</td>
<td>Releases beams for torsion.</td>
</tr>
</tbody>
</table>

**Figure 119: Space Frame File Generation dialog**
## Control

### Description

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Load Cases</strong></td>
<td></td>
</tr>
<tr>
<td><strong>group</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Control</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Wind Load</strong></td>
<td>Wind load will be applied to the analytical model using the load parameters indicated in the <strong>Wind Parameters</strong> dialog.</td>
</tr>
<tr>
<td><strong>Gust Load</strong></td>
<td>Gust (wind) load will be applied to the analytical model using the load parameters indicated in the <strong>Wind Parameters</strong> dialog.</td>
</tr>
<tr>
<td><strong>Temperature Load</strong></td>
<td>Note: This option is currently inactive.</td>
</tr>
<tr>
<td><strong>Seismic Load</strong></td>
<td>Earthquake Load per IS 1893 will be applied to the analytical model using the load parameters in the <strong>Seismic Parameters</strong> dialog.</td>
</tr>
</tbody>
</table>

### Building Type

This option is used to create appropriate load combinations.

- For **Irregular** buildings, both positive and negative direction load cases are generated.
- For **Regular** buildings, only positive direction load cases are generated.

### Generate EQ Load Using

Specifies that earthquake load are created using either **Reference Load** or by **Joint Weight**.

### Generate

Click generate the STAAD input file and open in Modeling mode.

### Related Links

- *M. To Generate a STAAD.Pro Model* (on page 923)

## M. To Design Slabs

You must have STAAD RCDC installed to perform slab design from the Building Planner mode.

You must perform a successful analysis of the generated STAAD.Pro input file prior to performing design.

1. Select **RCDC > Slab**.  
   STAAD RCDC opens and reads the analysis results for the associated STAAD.Pro input file.
2. In STAAD RCDC, select the slab and perform the design.

   **Tip:** Refer to the STAAD RCDC help for details.

3. Close STAAD RCDC when you have completed the slab designs for the selected building level.

## M. To Design Beams

You must have STAAD RCDC installed to perform beam design from the Building Planner mode.

You must perform a successful analysis of the generated STAAD.Pro input file prior to performing design.

1. Select **RCDC > Beams**.  
   STAAD RCDC opens and reads the analysis results for the associated STAAD.Pro input file.
2. In STAAD RCDC, select the beam and perform the design.

   **Tip:** Refer to the STAAD RCDC help for details.

3. Close STAAD RCDC when you have completed the beam designs for the selected building levels.
M. To Design Columns

You must have STAAD RCDC installed to perform column design from the Building Planner mode.

You must perform a successful analysis of the generated STAAD.Pro input file prior to performing design.

1. Select **RCDC > Columns**.
    STAAD RCDC opens and reads the analysis results for the associated STAAD.Pro input file.
2. In STAAD RCDC, select the column and perform the design.

    **Tip:** Refer to the STAAD RCDC help for details.

3. Close STAAD RCDC when you have completed the column designs.
This section of the help describes the analysis methods, data print commands, and how to perform analysis in STAAD.Pro

A. Types of Analysis

A. To specify a linear elastic analysis

To instruct the program to perform a first-order, linear elastic analysis of the structure, use the following procedure.

1. Either:
   - select the Analysis page
   or
   - on the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group

   The Analysis/Print Commands dialog opens.

2. Select the Perform Analysis tab.
3. Select the Print Option you want to use.
4. Click Add.
   The perform analysis command is added to the input file.

Related Links
- TR.37.1 Linear Elastic Analysis (on page 2796)
- Perform Analysis tab (on page 3065)

A. To specify a P-Delta analysis

To instruct the program to perform a second-order, P-Delta analysis, use the following procedure.

1. Either:
   - select the Analysis page
or

on the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group

The Analysis/Print Commands dialog opens.

2. Select the PDelta Analysis tab.

3. Specify a convergence option:
   type the Number of Iterations to use.

   or

   check the Converge option and then type the maximum number of iterations used to check convergence in the adjacent field, even if convergence has not been achieved

   **Note:** You should not specify a Number of Iterations when the Number of Iterations option is checked.

   You may leave the Converge option unchecked and leave the Number of Iterations as zero to use only a single iteration.

4. (Optional) Check the Use Geometric Stiffness (Kg) option to include stiffening effects of the KG matrix.

   Refer to Notes for Stress Stiffening Matrix (Option 2) (on page 2799) for details on this option.

5. Check the Small Delta option to include effects of member deflection (i.e., P-δ effects).

   This option is recommended. Clear this option to only consider effects of structure drift (i.e., P-Δ effects).

6. Select the Print Option you want to use.

7. Click Add.

   The PDetla Analysis command is added to the input file.

**Related Links**
- G.17.2.1 P-Delta Analysis (on page 2349)
- P Delta Analysis tab (on page 3066)
- TR.37.2 P-Delta Analysis Options (on page 2797)

**A. To specify a direct analysis**

To instruct the program to perform a direct analysis as per AISC 360, use the following procedure.

**Note:** You must have one or more members with Direct Analysis parameters defined in order to use a direct analysis.

This is a non-linear, iterative analysis as the stiffness of the members is dependent upon the forces generated by the load. The analysis will iterate in each step, changing the member characteristics until the maximum change in any τ₀ is less than the tau tolerance (τₜₒ₁). If the maximum change in any τ₀ is less than 100×τₜₒ₁ and the maximum change in any displacement degree of freedom is less than the displacement tolerance (δₜₒ₁); then the solution has converged for this case.

1. Either:

   select the Analysis page
or

on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

![Diagram]

The **Analysis/Print Commands** dialog opens.

2. Select the **Perform Direct Analysis** tab.

3. Select to use either the **LRFD** or **ASD** design method.

4. Type values for both **Tau** and **Displacement** Tolerances.

   **Note:** Leave either field empty to use the default tolerances of 0.01 for Tau and 0.01 inches and 0.01 radians for displacement.

5. Type the **Number of Iterations** to use.

   Leave this field empty (zero) to use the default of one iteration.

6. Select the number of **PDelta Iterations** to perform in the iterative PDelta with small delta analysis procedure with a direct analysis.

   The default value of 15 is recommended.

7. (Optional) Clear the **Reduced EI** option to use the full EI for member section moment and section displacement calculations.

8. (Optional) Clear the **Perform Tau-b Iteration** option prevent the program from iterating $\tau_b$.

9. Select the **Print Option** you want to use.

10. Click **Add**.

    The perform direct analysis command is added to the input file.

**Related Links**

- [G.17.2.1.4 AISC 360 Direct Analysis](#)
- [Perform Direct Analysis tab](#)
- [TR.37.5 Direct Analysis](#)

**A. To specify a nonlinear analysis**

To instruct the program to perform a geometric nonlinear analysis of the structure, use the following procedure.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

![Diagram]

   The **Analysis/Print Commands** dialog opens.

2. Select the **Nonlinear Analysis** tab.
3. (Optional) Specify the nonlinear analysis parameters as necessary:
   a. Type a absolute displacement limit for the first analysis step in the ARC field.
   b. Type the maximum number of Iterations to use to achieve equilibrium in the deform position within the specified tolerance value.
   c. Type the Tolerance to use for achieving convergence between two sequential iterations.
   d. Type the number of Load Steps to use for applying load in stages.
   e. Type frequency of Rebuild Steps of the tangent K matrix.
4. Check the KG option to add the geometric stiffness to the stiffness matrix.
5. (Optional) Check the Set Displacement Limit option to specify a target displacement:
   a. Either:
      type a Node number
   or
      click [...] to select a node in the view window
   b. Select a DOF (degree of freedom from the drop-down list).
   c. Type a Target Value distance for the displacement limit.
6. Select the Print Option you want to use.
7. Click Add. The perform nonlinear analysis command is added to the input file.

Related Links
- G.17.2.3 Static Geometrically Nonlinear Analysis (on page 2355)
- Nonlinear Analysis tab (on page 3071)
- TR.37.8 Geometric Nonlinear Analysis (on page 2830)
- TR.37.8 Geometric Nonlinear Analysis (on page 2830)
- TR.37.8 Geometric Nonlinear Analysis (on page 2830)
- Nonlinear Analysis tab (on page 3071)

A. To specify a nonlinear cable analysis

To specify a nonlinear cable analysis using the advanced features in STAAD.Pro, use the following steps.
You must specify at least one member with a nonlinear cable specification.

Note: This feature requires STAAD.Pro V8i (SELECTseries 5) (build 20.07.10) or higher.

1. Either:
   select the Analysis page
   or
   on the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group

The Analysis/Print Commands dialog opens.

2. Select the Perform Cable Analysis tab.
3. Select the **Advanced Cable Analysis** option.
4. (Optional) Set the options for **Use Modified Newton-Raphson** method and **Use Geometric Matrix (Kg)** as needed.
5. (Optional) Type values for the **Steps**, **Eq-Iterations**, and **Eq-tolerance** fields as needed. Default values are assumed for each when not specified.
6. Select the **Print Option** you want to use.
7. Click **Add**.
   The advanced cable analysis command is added to the input file.

**Tip:** You can select to add the cable sag values for a nonlinear cable analysis to the output file in the post-analysis **Analysis/Print Commands** dialog.

**Related Links**
- **Perform Cable Analysis tab** (on page 3067)
- **TR.37.3 Nonlinear Cable Analysis** (on page 2800)
- **Perform Cable Analysis tab** (on page 3067)

**A. To specify an imperfection analysis**

To instruct the program to perform an imperfection analysis, use the following procedure.

This analysis is used when member imperfection specifications are included in the model to define camber or drift.

1. Either:
   - select the **Analysis** page
   - or
   - on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Imperfection Analysis** tab.
3. Select the **Print Option** you want to use.
4. Click **Add**.
   The perform imperfection analysis command is added to the input file.

**Related Links**
- **G.17.2.4 Imperfection Analysis** (on page 2355)
- **TR.37.9 Imperfection Analysis** (on page 2832)
- **Perform Imperfection Analysis tab** (on page 3073)

**A. To specify buckling analysis**

To instruct the program to perform a buckling analysis on the structure, use the following procedure.
This analysis type is used to calculate the buckling factor for a load case. This is the load case multiplier at which global buckling of the structure would occur for that load case.

1. Either:
   
   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

The **Analysis/Print Commands** dialog opens.

2. Select the **Perform Buckling Analysis** tab.

3. Select the method for the buckling analysis:

   - Iterative method

   or

   - Eigen method - available for STAAD.Pro Advanced only

4. (Optional) If the iterative method is used, type the maximum **Number of Iterations** to use in the analysis.

   It not specified (i.e., left as zero), 20 iterations are used as the default value. 15 is recommended.

   **Note**: MAXSTEPS input is ignored when using the eigen method.

5. Select the **Print Option** you want to use.

6. Click **Add**.

   The perform buckling analysis command is added to the input file.

**Related Links**
- **G.17.2.2 Buckling Analysis** (on page 2353)
- **TR.37.4 Buckling Analysis** (on page 2804)
- **Perform Buckling Analysis tab** (on page 3074)

### A. To specify a pushover analysis

To instruct the program to perform a pushover analysis, use the following procedure.

You mush first define pushover data and the pushover loading.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group
The Analysis/Print Commands dialog opens.

2. Select the Perform Pushover Analysis tab.

3. Click Add.
   A message dialog opens to confirm you want to add this command.

Once you have successfully performed a pushover analysis, you can review the pushover results in the Postprocessing workflow.

Related Links
- TR.37.7 Pushover Analysis (on page 2819)
- G.17.4 Pushover Analysis (on page 2376)
- M. Pushover Loads (on page 861)
- Perform Pushover Analysis tab (on page 3075)

A. To add a change command

To add a change command which is required for performing a subsequent analysis command, use the following procedure.

A change command is only added after an analysis command.

In addition to being used for a new analysis, the change command is used to change the support conditions, active/inactive member status, load conditions, etc. The stiffness matrix being solved will be reset when a change command is used.

1. Either:
   - select the Analysis page
   - or
   - on the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group

   The Analysis/Print Commands dialog opens.

2. Select the Change tab.

3. (Optional) Check the After Current option if you previously selected the analysis command in the input file list within the Analysis - Whole Structure dialog.
   This option allows you to specify specifically where the command is added. Otherwise it will be added to the end of the input file.

4. Click Add.
   The change command is added to the input file.

You can now add an additional analysis command to the input file. A change command with no subsequent analysis command has no effect.

Related Links
- G.19 Multiple Analyses (on page 2401)
- TR.38 Change Specification (on page 2835)
- Change tab (on page 3069)
A. To generate a floor spectrum

To calculate the floor spectra from time history acceleration, use the following procedure.

This procedure should only be used if a time history load case is being solved by the analysis engine.

**Note:** The floor response spectrum command must immediately follow an analysis command. Therefore, it is helpful to select the corresponding analysis command prior to starting this procedure.

**Note:** That analysis can only contain a single time history load case.

This command is used to specify the calculation of floor and/or joint spectra from time history results.

1. Either:
   - select the Analysis page
   - or
   - on the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group

   The Analysis/Print Commands dialog opens.

2. Select the Generate Floor Spectrum tab.

3. For every group and direction that is necessary:
   a. Check one or more global directions in a table row: GX, GY, and GZ.
   b. Type a Title used to identify the spectrum.
   c. Use the drop-down to select the Node Groups which are included in this direction.

   You may include as many directions as needed.

4. (Optional) Type frequency options to use for the spectra (all are optional inputs):
   a. Type the Lowest Frequency (in Hz) to be in the calculated spectrum.
      This should be at least 0.0 Hz.
   b. Type the Highest Frequency (in Hz) to be in the calculated spectrum
   c. Type the number of Frequency Interval to use between the lowest and highest frequency.

5. (Optional) Type up to ten Damping Ratios to use as a ratio.
   (e.g., 3% damping is 0.03)

   One spectrum will be generated for each damping ratio for each global direction requested for each floor defined. The spectrum will be based on these damping ratios. Is not entered, the default of 5% (0.05) damping will be used as a single damping ratio.

6. Check the Relative Acceleration? option if there is ground motion defined and you want the spectra based on the relative acceleration of the floor to the ground acceleration.
   Otherwise, the acceleration is assumed to be an absolute acceleration.

7. (Optional) Select the print parameters as needed (both are optional):
   a. Check the Print Time History Acceleration used? option to include this in the output.
   b. Check the Print Calculated Spectrum? option to include this in the output.

8. Select the Print Option you want to use.
Analysis
A. To specify pre-analysis commands

9. Click Add.

Tip: The After Current option is helpful to make sure this is added immediately following the analysis command if you selected prior to this procedure.

The generate floor spectrum command is added to the input file.

Related Links
• TR.37.10 Floor Spectrum Command (on page 2833)
• TR.37.10 Floor Spectrum Command (on page 2833)
• Generate Floor Spectrum tab (on page 3070)

A. To specify pre-analysis commands

To instruct the program to include model input data in the STAAD output file (.an1), use the following procedure.

1. On the Analysis and Design ribbon tab, select the Pre Analysis Commands tool in the Analysis Data group.

The Pre Analysis Print - Whole Structure dialog opens.

2. Click Define Commands.

The Analysis/Print Commands dialog opens.

3. Select the tab corresponding to the print command you want to add to the output file.

   Problem Statistics
   Joint Coordinates
   Member Information
   Material Properties
   Support Information
   All
   Element Information
   Solid Information
   Member Properties

4. Click Add.

Note: If you selected a portion of the model prior to opening the Analysis/Print Commands dialog, you can click Assign to assign the command to those model objects.

The selected print command is added to the input file.

5. Repeat Steps 3 and 4 to add additional print commands.

6. Click Close.

You must assign most print commands to portions of the model using one of the standard assignment methods.

Related Links
• TR.42 Print Specifications (on page 2840)
**A. To create a load list**

To create an active list of load cases for use with subsequent print and design commands, use the following procedure.

A load list is used to control the loads that are included with following print and design specifications.

*Note:* All load cases are considered by analysis commands. You just a CHANGE command to make changes to the loads considered for analysis.

1. On the **Analysis and Design** ribbon tab, select the **Load List** tool in the **Analysis Data** group.

   The **Load List** dialog opens.

2. Either:
   
   select one or more items in the **Load Cases** list and then click **[>]** to include them in the **Load List**
   
   or
   
   click **[>>]** to include all load cases

   *Tip:* Including all load cases is the same as not using a load list at all.

3. Click **OK**

**Related Links**

- **Load List dialog** (on page 3080)
- **TR.39 Load List Specification** (on page 2836)

**A. To check for soft stories and seismic code irregularities**

To check for soft stories and other seismic code irregularities in rigid diaphragm structures, use the following procedure.

*Note:* Soft story checks and plan irregularities are applicable to structures with rigid floor diaphragm definitions. Refer to **M. To assign nodes to a floor diaphragm** (on page 811).

The **CHECK SOFT STORY** option may added to a **FLOOR DIAPHRAGM** command set to check for soft stories per either IS 1893 2002, IS 1893 2016, or ASCE 7-05 codes. The **CHECK IRREGULARITIES** command may be added to check plan irregularities for the IS 1893 2016 code.

*Tip:* You can check plan and vertical irregularities and structure stiffness per EC8 using the **Earthquake workflow** (on page 2240).

1. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands > Floor Diaphragm Options** tool in the **Analysis Data** group.
The **Floor Diaphragm Options** dialog opens.

2. Select the building code for which you want to perform earthquake checks:
   - IS1893 2002
   - IS1893 2016
   - ASCE7-05

3. Check the option **Check Soft Story** to perform this check.

4. (IS 1893 - 2016 only) To check for plan and other vertical irregularities, check the option **Check Irregularities**.
   Horizontal irregularities (torsional and reentrant corners) per Table 5 and vertical irregularities (mass irregularities and irregular modes of oscillation) per Table 6 of IS 1893 2016 can be checked by the program using this command.

5. (Optional) If the base of the structure is not the minimum global Y coordinate defined in the structure, then check the **Base Level** option and then type the height of the structure base in the adjacent field.

6. Click **Add**.

**Note:** Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command `PRINT STORY STIFFNESS` in the post analysis print commands section.

**Related Links**
- **TR.28.2 Floor Diaphragm** (on page 2526)
- **TR.28.2.1 Soft Story Checking** (on page 2529)
- **Floor Diaphragm Options dialog** (on page 3079)
A. To specify post-analysis print commands

To include additional analysis results in the STAAD output (.anl) file, use the following procedure.

**Note:** All output values will be available in the postprocessing workflow, regardless of whether you add them to the output file. Similarly, you can add post-analysis output to a report even if not included in the output file.

1. Select the **Post Analysis Commands** tool in the **Analysis Data** group on the **Analysis and Design** ribbon tab.

   ![Analysis/Print Commands](image)

   The **Analysis/Print Commands** dialog for post-analysis commands opens.

   **Tip:** If the dialog does not immediately open, click **Define Commands** on the **Post Analysis Print - Whole Structure** dialog.

2. Select the tab pertaining to the output you want to add to the output file.

3. Check the **After Current** option to insert the print command following the currently selected command within the **Post Analysis Print - Whole Structure** dialog.
   
   Otherwise, the print command will be added to the end of the STAAD input file.

4. Click **Add**.

   The command is added to the STAAD Input file.

5. (Optional) Repeat steps 2 through 4 to add additional print commands as needed.

6. Click **Close**.

   **Note:** Some print commands require a assignment. These will be marked with question mark (?) in the **Post Analysis Print - Whole Structure** dialog. Use the **Assignment Method** tools to assign these to the appropriate model objects.

**Related Links**

- **Analysis/Print Commands dialog (Post Print)** (on page 3076)
- **TR.42 Print Specifications** (on page 2840)
A. To output the center of rigidity

To instruct the program to include the center of rigidity of each rigid diaphragm in the analysis output, use the following procedure.

The lateral force at each floor, as generated by earthquake and wind loading, acts at the center of rigidity of each floor which is modeled as rigid floor diaphragm. The center of mass of each floor is defined as the mean location of the mass system of each floor. The mass of the floor is assumed to be concentrated at this point when the floor is modeled as rigid diaphragm. The distance between these two is the lever arm for the natural torsion moment for seismic loads when that option is used.

1. Select the Post Analysis Commands tool in the Analysis Data group on the Analysis and Design ribbon tab.

   ![Post Analysis Commands](image)

   The Analysis/Print Commands dialog for post-analysis commands opens.

   **Tip:** If the dialog does not immediately open, click Define Commands on the Post Analysis Print - Whole Structure dialog.

2. Select the Dia CR tab in the Analysis/Print Commands dialog.

3. Check the After Current option to insert the print command following the currently selected command within the Post Analysis Print - Whole Structure dialog.

   Otherwise, the print command will be added to the end of the STAAD input file.

4. Click Add.

   The command is added to the STAAD Input file.

5. Click Close.

**Related Links**

- [TR.42 Print Specifications](on page 2840)

A. To report cable sag from an advanced cable analysis

To include the cable sag in the output from an advanced cable analysis, use the following procedure.

**Tip:** Cable sag is only available for advanced nonlinear analysis. If requested for a standard cable analysis, a warning message will be reported in the output.

An advanced cable analysis can also report the deflected shape of the sag in the cable using the PRINT CABLE SAG command.

1. On the Analysis and Design ribbon tab, select the Post Analysis Commands tool in the Analysis Data group.

   ![Post Analysis Commands](image)
A. To specify post-analysis print commands

1. Select the Post Analysis Print - Whole Structure dialog opens.
2. Click Define Commands in the Post Analysis Print - Whole Structure dialog.
3. Select the Analysis/Print Commands dialog opens.
4. Click Add.
5. Click Add.
6. Enter an Allowable Drift Factor value.
7. This is the drift ratio value as specified by the applicable building code.
8. Check the After Current option to insert the print command following the currently selected command within the Post Analysis Print - Whole Structure dialog.
9. Otherwise, the print command will be added to the end of the STAAD input file.
10. Click Add.

The output of a successful advanced cable analysis will report the cable sag in local XYZ coordinates. Post-analysis print will calculate the actual, nonlinear cable displacements along the length of cable.

A. To check for inter-story drift

To instruct the program to check the drift between adjacent stories against a code-defined ratio, use the following procedure.

Story drift is calculated as the relative horizontal displacement of two adjacent floors in a building. Inter-story drift can also be expressed as some factor times the story height. The allowable factor generally varies from one country code to another and may also vary depending on the type of loading. For example, in IS 1893: 2002 for seismic loading the allowable limit for inter-story drift is 0.004 times the story height whereas in IS 875 for wind loading it is 0.002 times the story height.

The drift at a particular story level in either lateral direction is calculated as the average of all the joint displacements present in that floor level. However if floor diaphragm is present, the drift is calculated at the center of mass (i.e., master joint) of the floor.

Note: Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command PRINT STORY STIFFNESS in the post analysis print commands section.

Note: For dynamic IS 1893: 2002 response spectrum, the story drift check is performed by adding a command line within the load case, rather than in the post-analysis print commands. Refer to TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2721) for additional information.
The command is added to the STAAD Input file.

6. Click Close.

Related Links
• TR.42 Print Specifications (on page 2840)

A. To perform an analysis in STAAD.Pro

Once you have completed the input file, use the following procedure to perform an analysis and optional design.

The STAAD analysis engine performs analysis and design sequentially with a single click. In order to carry out the design, these design parameters must be specified along with geometry, properties, etc. in the input file (this is referred to as a “batch” design). Also, note that you can change the design code used for design and code check before performing the analysis and design.

The Analytical Modeling and Physical Modeling of the STAAD.Pro user interface are used to prepare the structural input data which is then passed to the STAAD analysis engine for general purpose structural analysis and design.

1. Either:
   - Select the Run Analysis tool in the Analysis group on the Analysis and Design ribbon tab.

or

Press <CTRL+F5>

The STAAD Analysis and Design dialog opens.

During the analysis (and design, if specified), an output file is generated. This file may contain selected input data items, results and error messages. Optional print specifications can be used to include additional information in the output file.

2. Select an option for what action occurs when the dialog is closed:
   - Open the output file
   - go to the post-processing mode
   - remain in the analytical modeling mode

3. Click Done.

4. (Optional) To review the output file is this option was not selected in the STAAD Analysis and Design dialog, either:
   - Select the STAAD Output tool in the Utilities group on the Utilities ribbon tab
Analysis

A. To perform an analysis in STAAD.Pro

or

Select the **View STAAD Output File** tool on the Quick Start toolbar.

**Related Links**

- *STAAD Analysis and Design dialog* (on page 3080)
- *STAAD Analysis and Design dialog* (on page 3080)
This section contains information on designing structural elements, organized by material type.

D. Batch Design versus Interactive Design Workflows

STAAD.Pro has two means by which structural members can be designed.

Batch Design

Using this method, code checks and/or member selection is performed directly by the analysis and design engine when an analysis is performed.

Interactive Design Workflow

Code checks and member selection are performed in post-processing workflow for interactive Concrete Design. This workflow is available in the Workflows panel.

Related Links

- *D. Available Steel Design Codes* (on page 944)
- *D. Available Concrete Design Codes* (on page 1000)
- *D. Available Aluminum Design Codes* (on page 1362)
- *D. Available Timber Design Codes* (on page 1364)

D. Steel Design

D. Available Steel Design Codes

All steel design codes are available using the batch design method in the Analytical Modeling workflow (e.g., via the STAAD input file) where not specifically stated otherwise in the workflow column.
## Batch Design

**Table 47: Steel design codes available in batch design**

<table>
<thead>
<tr>
<th>Country/Region</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Australia</td>
<td>AS 4100-1998 (on page 1633)</td>
<td>Standards Australia - Steel Structural Design, including Amendment 1 (2012)</td>
</tr>
<tr>
<td>Canada</td>
<td>CAN/CSA-S16-01 (on page 1701)</td>
<td>Limit States Design of Steel Structures</td>
</tr>
<tr>
<td></td>
<td>S136-94 (on page 1715)</td>
<td>Specification for the Design of Cold-Formed Steel Structural Members</td>
</tr>
<tr>
<td></td>
<td>CAN/CSA-S16-09/14 (on page 1728)</td>
<td>Limit States Design of Steel Structures (2009 and 2014 editions)</td>
</tr>
<tr>
<td>Finland</td>
<td>SFS EN 1993-1-1 (on page 1820)</td>
<td>Finnish National Annex (EC3)</td>
</tr>
<tr>
<td>France</td>
<td>CM66 1977 (on page 1872)</td>
<td>Regulations for the calculation of steel structures and Design rules for structural steelwork, Addendum 80</td>
</tr>
<tr>
<td></td>
<td>NF EN 1993-1-1/NA (on page 1813)</td>
<td>French National Annex (EC3)</td>
</tr>
<tr>
<td>Germany</td>
<td>DIN 18 800 Part 1 (on page 1880)</td>
<td>Structural steelwork. Safety against buckling of linear members and frames</td>
</tr>
<tr>
<td></td>
<td>DIN EN 1993-1-1:2005 (on page 1849)</td>
<td>German National Annex (EC3)</td>
</tr>
<tr>
<td>Country/Region</td>
<td>Code</td>
<td>Title</td>
</tr>
<tr>
<td>---------------</td>
<td>-------------------------------</td>
<td>----------------------------------------------------------------------</td>
</tr>
<tr>
<td>India</td>
<td>IS 800 1984, ASD (on page 1916)</td>
<td>Code of Practice for General Construction in Steel</td>
</tr>
<tr>
<td></td>
<td>IS 800 2007, LSD (on page 1942)</td>
<td>General Construction in Steel - Code of Practice (Third Revision)</td>
</tr>
<tr>
<td></td>
<td>IS 800 2007, WSD (on page 1942)</td>
<td>General Construction in Steel - Code of Practice (Third Revision)</td>
</tr>
<tr>
<td></td>
<td>IS 801 1975 (on page 1930)</td>
<td>Code of Practice for Use of Cold-Formed Light Gauge Steel Structural Members in General Build Construction</td>
</tr>
<tr>
<td>Japan</td>
<td>AIJ 2002 (on page 1988)</td>
<td>Design Standard for Steel Structures</td>
</tr>
<tr>
<td></td>
<td>AIJ 2005 (on page 1976)</td>
<td>Design Standard for Steel Structures</td>
</tr>
<tr>
<td>Malaysia</td>
<td>MS-EN 1993-1-1 (on page 1844)</td>
<td>Malaysian National Annex (EC3)</td>
</tr>
<tr>
<td>Mexico</td>
<td>NTC 1987 (on page 2013)</td>
<td>Normas Técnicas Complementarias para Diseño y construcción de Estructuras Metálicas - LRFD</td>
</tr>
<tr>
<td>Netherlands</td>
<td>NEN EN 1993-1-1/NB (on page 1794)</td>
<td>Dutch National Annex (EC3)</td>
</tr>
<tr>
<td>New Zealand</td>
<td>NZS 3404-1997 (on page 2021)</td>
<td>New Zealand Standard for Steel Structures, Parts 1 &amp; 2, including Amendments 1 &amp; 2.</td>
</tr>
<tr>
<td>Country/Region</td>
<td>Code</td>
<td>Title</td>
</tr>
<tr>
<td>---------------</td>
<td>------</td>
<td>-------</td>
</tr>
<tr>
<td>NORSOK N-004</td>
<td>NORSOK N-004 Rev 2, October 2004</td>
<td></td>
</tr>
<tr>
<td>Russia</td>
<td>SNiP 2.23-81* 1990 (on page 2137)</td>
<td>Design Standards for Steel Construction</td>
</tr>
<tr>
<td></td>
<td>SP 16.13330.2011 (on page 2152)</td>
<td>Steel Structures</td>
</tr>
<tr>
<td>Poland</td>
<td>PN-EN 1993-1-1 (on page 1826)</td>
<td>Polish National Annex (EC3)</td>
</tr>
<tr>
<td></td>
<td>SANS 10162-1:2011 (on page 2200)</td>
<td>Design of steel structures</td>
</tr>
<tr>
<td>Sweden</td>
<td>BFS EN 1993-1-1:2005 (on page 1853)</td>
<td>Swedish National Annex (EC3)</td>
</tr>
<tr>
<td>United Kingdom</td>
<td>BS 5950-1:2000 (on page 1655)</td>
<td>Structural use of steelwork in building - Part 1: Code of practice for design - Rolled and welded sections, Incorporating Corrigendum No. 1</td>
</tr>
<tr>
<td></td>
<td>BS 5950-5:1998 (on page 1682)</td>
<td>Structural use of steelwork in building - Part 5. Code of practice for design of cold formed thin gauge sections</td>
</tr>
<tr>
<td>Country/Region</td>
<td>Code</td>
<td>Title</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>--------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td><strong>BS EN 1993-1-1:2005</strong> (on page 1804)</td>
<td>British National Annex (EC3)</td>
</tr>
<tr>
<td>Country/Region</td>
<td>Code</td>
<td>Title</td>
</tr>
<tr>
<td>---------------</td>
<td>----------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>AISC ASD 1989 (on page 1422)</td>
<td>AISC ASD 1989 9th edition Castellated</td>
</tr>
<tr>
<td></td>
<td>AISI 1996 (on page 1473)</td>
<td>Design of Cold Formed Steel Structural Members</td>
</tr>
<tr>
<td></td>
<td>AASHTO ASD (on page 1464)</td>
<td>Standard Specifications for Highway Bridges, 17th Edition (2002); Chapter 10, Part C</td>
</tr>
<tr>
<td></td>
<td>ASCE 10-97 (on page 1553)</td>
<td>Design of Latticed Steel Transmission Structures</td>
</tr>
</tbody>
</table>

### Nuclear Codes

**Note:** These codes require a separate Nuclear Design Codes license.

<table>
<thead>
<tr>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASME NF 3000 1974 (on page 1588)</td>
<td>ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF</td>
</tr>
</tbody>
</table>
### Design

**D. Steel Design**

<table>
<thead>
<tr>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>ASME NF 3000 1977</strong> (on page 1588)</td>
<td>ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF</td>
</tr>
<tr>
<td><strong>ASME NF 3000 1989</strong> (on page 1597)</td>
<td>ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF</td>
</tr>
<tr>
<td><strong>ASME NF 3000 1998</strong> (on page 1607)</td>
<td>ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF</td>
</tr>
<tr>
<td><strong>ASME NF 3000-2001</strong> (on page 1616)</td>
<td>ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF</td>
</tr>
<tr>
<td><strong>ASME NF 3000-2004</strong> (on page 1616)</td>
<td>ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF</td>
</tr>
<tr>
<td><strong>AISC N690 1994-s1</strong> (on page 1575)</td>
<td>Specification of the Design, Fabrication and Erection of Steel Safety-Related Structures for Nuclear Facilities and Supplement No. 1</td>
</tr>
</tbody>
</table>

**Related Links**

- *D. To specify steel design code and parameters* (on page 951)
- *D. Batch Design versus Interactive Design Workflows* (on page 944)

### D. Batch Steel Design Operations

STAAD.Pro contains a broad set of facilities for designing structural steel members as individual components of an analyzed structure. The member design facilities provide you with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem.

The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.
- Specify design parameter values, if different from the default values.

You may repeat these operations any number of times depending upon the design requirements.

For design per American steel codes, STAAD.Pro supports steel design of wide flange, S, M, HP shapes, tees, angle, double angle, channel, double channel, pipes, tubes, beams with cover plate and composite beams (I shapes with concrete slab on top). For other steel design codes, a wide variety of steel shapes are supported.
Refer to the D. Design Codes (on page 1366) for details on the member property specifications and supported steel shapes for design for other codes.

D. To specify steel design code and parameters

To initiate the design of steel members and specify the code parameters, use the following procedure.

Note: To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

2. In Steel Design - Whole Structure dialog, select the applicable steel design code from the Current Code drop-down list.
3. Click Define Parameters. The Design Parameters dialog opens.
4. Specify a value or option for each required parameter for a set of members and then click Add. You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.

Note: Different parameters can be used for different member type designs (e.g., columns, beams, composite floor girders, braces, etc.). Alternatively, you can use a separate set of parameters for different member types.

5. Click Close. The design parameters are marked with an icon. This indicates that the need to be assigned to members.

6. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.

Related Links
- D. Available Steel Design Codes (on page 944)
- Steel Design - Whole Structure dialog (on page 3081)
- Design Parameters dialog (on page 3083)
- TR.48.1 Parameter Specifications (on page 2851)

D. To design steel members in groups

To design members as a group, use the following procedure.

In many situations, it is preferable to have only a few different member sizes for many members rather than each member size be different, even if this would result in some members being underutilized. You can group members in the batch design to instruct the program to replace the member sizes in the group with the highest capacity member of the existing sizes in that group. You can also instruct the program to use a specified section property for the grouping or even group based on a specific member number, regardless of property specifications.

For example, in a floor bay with 5 beam members the edge members were selected as W12x14 shapes and the interior members were selected as W14x22 shapes. Grouping these members will immediately instruct the program to use W14x22 for all members for any subsequent instructions (i.e., analysis, design, take-off, etc.).
Note: All members within a group must be of the same cross section type. If one or more of the members has a different cross-section type, the grouping is ignored by the program.

1. On the Steel Design - Whole Structure dialog, click Commands.
   The Design Commands dialog opens. The tabs (if any) available in this dialog are dependant on the current design code selection. Not all commands are available for every design code.

2. If you want to retain the member groups for all subsequent commands, select the FIXED GROUP tab and then click Add.
   If you do not use the FIXED GROUP option, then the members are grouped once and then this grouping is ignored for subsequent commands. If you do add the FIXED GROUP command, then all subsequent member selection commands will apply to the group again.

3. Select the GROUP tab.

4. (Optional) Select the Property Specification to use for grouping shapes.

5. (Optional) Check the Same as Beam # option and then select a member number from the drop-down list.

6. Click Add.

7. Click Close.

8. Assign the group command to all the members you want to group together.

You can repeat this procedure to create as many design groups as necessary.

Once you have created a group, you will typically want to perform a code check or member selection on the members.

Related Links
- TR.48 Steel and Aluminum Design Specifications (on page 2850)
- Design Commands dialog (on page 3084)
- Steel Design - Whole Structure dialog (on page 3081)
- TR.50 Group Specification (on page 2855)

D. To specify steel design commands

To specify the code checking or design commands to be used for the steel design operation, use the following procedure.

1. On the Steel Design - Whole Structure dialog, click Commands.
   The Design Commands dialog opens. The tabs (if any) available in this dialog are dependant on the current design code selection. Not all commands are available for every design code.

2. To perform a design action, select one of the following options (as they are available for the code selection):
   - To... Set...
     - check the capacity of the current member sizes the CHECK CODE tab
     - select a member size based on the last analysis results the SELECT tab
     - select a member size based on iterating the analysis for updated member sizes the SELECT OPTIMIZED tab

3. Click Add.

4. Click Close.

5. Assign the design commands to the appropriate members.

Related Links

STAAD.Pro 952 User Manual
D. To generate steel take off

To generate a summary of all steel sections with their lengths and weights for the whole structure or a list of members, use the following procedure.

1. On the Steel Design - Whole Structure dialog, click Commands.
   The Design Commands dialog opens. The tabs (if any) available in this dialog are dependant on the current design code selection. Not all commands are available for every design code.
2. Select the take off table you want to include:
   
   To... | Select...
   --- | ---
   generate a list of sections used including total length and weight | the TAKE OFF tab
   generate a list sorted by members including the length and weight of the member and section used | the MEMBER TAKE OFF tab

3. Click Add.
4. Click Close.
5. Assign the take off commands to members or to a named group of members.

Related Links
- TR.51 Steel and Aluminum Take Off Specification (on page 2856)
- Design Commands dialog (on page 3084)
- P. Steel AutoDrafter Workflow (on page 2227)

D. Steel Connection Design

D. Connection Design workflow

D. Getting Started

The following concepts will help you understand how to use Connection Design in STAAD.Pro for the design of steel connections in your engineering practice.

D. Overview

Purpose

The Connection Design mode in STAAD.Pro is used to dynamically link structural model data —including section properties and analysis results— to the Connection Design application in order to check connection designs for code compliance. The resulting data and diagrams of the connection can also be included in the User Report.
**Note:** Not all options described here will be available for versions of Connection Design prior to release 7.0.

**Note:** Full use of the Connection Design mode in STAAD.Pro requires a valid RAM Connection license. The RAM Connection mode will run with a limited set of steel connections available with only a standard STAAD.Pro license. Please contact your Bentley account manager to have a Connection Design license added to your SELECT licenses.

**Description**

Connections are designed in the Connection Design Mode by creating "Joints" from the geometry and section properties. Forces resulting from the analysis are used in this mode by assigning a load envelope. A set of connection templates is then assigned to the joint. A suitable connection design, if one is available, will be reported once you have selected the appropriate connection templates.

Any number of joints may be selected and designed. Further, connection design is performed automatically for you once appropriate templates have been selected for the selected joints. These enhancements greatly reduce the time required for connection design in models of all sizes.

**Tip:** The selected load envelope is now used for all connection designs, instead of per design brief as in previous versions of STAAD.Pro.

Refer to the following procedures and interface elements used in designing steel connections with RAM Connection within STAAD.Pro. For specific information regarding RAM Connection templates, refer to the documentation with RAM Connection.

**D. Full vs. Free Connection Sets**

Basic connection templates are available to users with a valid STAAD.Pro license. The following connections templates are available when using a standard STAAD.Pro license.

- Basic DA BCF Bolted
- Basic DA BCF Welded
- Basic EP BCF Bolted
- Basic EP BCF Welded
- Basic DA BCW Welded
- Basic EP BCW Welded
- Basic SP BCF
- Basic SP BCW

**Note:** In order to use all connection templates (including Smart and Gusset connections), a valid RAM Connection license is required.

For a detailed explanation of connections available, refer to AD.2007-1001.5.1 in the What's New section.

**D. Allowable Member Types Per Connection**
<table>
<thead>
<tr>
<th>Connection</th>
<th>Family</th>
<th>Member Types</th>
<th>Built-up Symmetric</th>
<th>Built-up Nonsymmetric</th>
<th>Member Flange Rotation</th>
<th>Sections Allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base plate</td>
<td>CB</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I, I2C, HSS_RECT, HSS_CIRC</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>CB</td>
<td>Tapered</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Bent plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Bracket</td>
<td>Bracket plate</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>Tee bracket</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>T</td>
</tr>
<tr>
<td>BS4Angles</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Cap plate</td>
<td>CP</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td>Clip angle</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Directly welded</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td>End plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Flange plate</td>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Top Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td>Connection</td>
<td>Family</td>
<td>Member Types</td>
<td>Built-up Symmetric</td>
<td>Built-up Nonsymmetric</td>
<td>Member Flange Rotation</td>
<td>Sections Allowed</td>
</tr>
<tr>
<td>------------</td>
<td>--------</td>
<td>--------------</td>
<td>--------------------</td>
<td>-----------------------</td>
<td>------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
<td></td>
</tr>
<tr>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
<td></td>
</tr>
<tr>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>Gusset base plate</td>
<td>GBP</td>
<td>Prismatic</td>
<td>YES</td>
<td>Column &quot;YES&quot; - Braces &quot;NO&quot;</td>
<td>Column &quot;YES&quot;</td>
<td>Column &quot;I, HSS_RECT&quot; - Braces &quot;I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td>Gusset chevron</td>
<td>CVR</td>
<td>Prismatic</td>
<td>YES</td>
<td>BEAM &quot;YES&quot; - Braces &quot;NO&quot;</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Braces &quot;I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td>Gusset column beam brace</td>
<td>CBB</td>
<td>Prismatic</td>
<td>YES</td>
<td>Column &quot;YES&quot; - Beams &quot;YES&quot; - Braces &quot;NO&quot;</td>
<td>Column &quot;YES&quot;</td>
<td>Column &quot;I, HSS_RECT&quot; - Beam &quot;I&quot; - Braces &quot;I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td>Gusset VXB</td>
<td>VXB</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Braces &quot;I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td>Moment end plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>BS</td>
<td>Tapered</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>BS</td>
<td>Haunched</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>1, T</td>
<td></td>
</tr>
<tr>
<td>BCF</td>
<td>Column, Beam &quot;Prismatic&quot;</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
<td></td>
</tr>
<tr>
<td>Connection</td>
<td>Family</td>
<td>Member Types</td>
<td>Built-up Symmetric</td>
<td>Built-up Nonsymmetric</td>
<td>Member Flange Rotation</td>
<td>Sections Allowed</td>
</tr>
<tr>
<td>--------------------</td>
<td>--------</td>
<td>--------------</td>
<td>--------------------</td>
<td>-----------------------</td>
<td>------------------------</td>
<td>----------------------------------</td>
</tr>
<tr>
<td>BCF</td>
<td>Tapered &quot;Beam&quot;</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Beam &quot;I&quot;</td>
<td></td>
</tr>
<tr>
<td>BCF</td>
<td>Haunched &quot;Beam&quot;</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Haunch &quot;I, T&quot;</td>
<td></td>
</tr>
<tr>
<td>BCW</td>
<td>Column, Beam &quot;Prismatic&quot;</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;HSS_RECT&quot;</td>
<td></td>
</tr>
<tr>
<td>BCW</td>
<td>Tapered &quot;Beam&quot;</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot;</td>
<td></td>
</tr>
<tr>
<td>BCW</td>
<td>Haunched &quot;Beam&quot;</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Haunch &quot;I, T&quot;</td>
<td></td>
</tr>
<tr>
<td>Moment end plate Knee</td>
<td>BCF</td>
<td>Tapered</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td>PRConnector</td>
<td>PR</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Single plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT, HSS_CIRC&quot;</td>
<td></td>
</tr>
<tr>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT, HSS_CIRC&quot;</td>
<td></td>
</tr>
<tr>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
<td></td>
</tr>
<tr>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Top Column &quot;YES&quot;</td>
<td>I</td>
<td></td>
</tr>
<tr>
<td>Standard tee</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSSRECT&quot;</td>
</tr>
<tr>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
<td></td>
</tr>
<tr>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I</td>
<td></td>
</tr>
<tr>
<td>Connection</td>
<td>Family</td>
<td>Member Types</td>
<td>Built-up Symmetric</td>
<td>Built-up Nonsymmetric</td>
<td>Member Flange Rotation</td>
<td>Sections Allowed</td>
</tr>
<tr>
<td>------------------</td>
<td>--------</td>
<td>--------------</td>
<td>--------------------</td>
<td>-----------------------</td>
<td>------------------------</td>
<td>-------------------------------------------------------</td>
</tr>
<tr>
<td>Stiffened seated</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td>Through plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td>Unstiffened seated</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
</tbody>
</table>

Table 49: BS5950

<table>
<thead>
<tr>
<th>Connection</th>
<th>Family</th>
<th>Member Types</th>
<th>Built-up Symmetric</th>
<th>Built-up Nonsymmetric</th>
<th>Member Flange Rotation</th>
<th>Sections Allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bolted end plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>BS</td>
<td>Tapered</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>BS</td>
<td>Haunched</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I, T</td>
</tr>
<tr>
<td>Cleat angle</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>Connection</td>
<td>Family</td>
<td>Member Types</td>
<td>Built-up Symmetric</td>
<td>Built-up Nonsymmetric</td>
<td>Member Flange Rotation</td>
<td>Sections Allowed</td>
</tr>
<tr>
<td>------------------</td>
<td>--------</td>
<td>--------------</td>
<td>--------------------</td>
<td>-----------------------</td>
<td>------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>Fully welded</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>Flanges welded</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>Fin plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I&quot;, HSS_RECT, HSS_CIRC</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I&quot;, HSS_RECT, HSS_CIRC</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>Flange cover plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>Flexible end plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I&quot;, HSS_RECT</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I&quot;, HSS_RECT</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>Moment end plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I&quot; - Hauch &quot;I, T&quot;</td>
</tr>
<tr>
<td></td>
<td>BCF</td>
<td>Haunched</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I&quot; - Hauch &quot;I, T&quot;</td>
</tr>
<tr>
<td>Web cover plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>1</td>
</tr>
</tbody>
</table>
## Table 50: GB 50017-03

<table>
<thead>
<tr>
<th>Connection</th>
<th>Family</th>
<th>Member Types</th>
<th>Built-up Symmetric</th>
<th>Built-up Nonsymmetric</th>
<th>Member Flange Rotation</th>
<th>Sections Allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base plate</td>
<td>CB</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Column &quot;I, I2C, HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tapered YES NO NO I</td>
</tr>
<tr>
<td>Clip angle</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCW YES NO NO</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BG YES NO NO</td>
</tr>
<tr>
<td>Directly welded</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCW YES NO NO</td>
</tr>
<tr>
<td>Flange plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCW YES NO NO</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BG YES NO NO</td>
</tr>
<tr>
<td>Moment end plate</td>
<td>BCF</td>
<td>Column, Beam &quot;Prismatic&quot;</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td></td>
<td>BCF</td>
<td>Tapered &quot;Beam&quot;</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot;</td>
</tr>
<tr>
<td></td>
<td>BCF</td>
<td>Haunched &quot;Beam&quot;</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Haunch &quot;I, T&quot;</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Column, Beam &quot;Prismatic&quot;</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Beam &quot;I&quot; - Column &quot;HSS_RECT&quot;</td>
</tr>
</tbody>
</table>
### Table 51: EN 1993-2005

<table>
<thead>
<tr>
<th>Connection</th>
<th>Family</th>
<th>Member Types</th>
<th>Built-up Symmetric</th>
<th>Built-up Nonsymmetric</th>
<th>Member Flange Rotation</th>
<th>Sections Allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base plate</td>
<td>CB</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>Column &quot;I, I2C, HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td>Bolted end plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BS</td>
<td>Haunched</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I, T</td>
</tr>
<tr>
<td></td>
<td>BCF</td>
<td>Column, Beam &quot;Prismatic&quot;</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCF</td>
<td>Haunched</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Haunch &quot;I, T&quot;</td>
</tr>
<tr>
<td>Cleat angle</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td>Connection</td>
<td>Family</td>
<td>Member Types</td>
<td>Built-up Symmetric</td>
<td>Built-up Nonsymmetric</td>
<td>Member Flange Rotation</td>
<td>Sections Allowed</td>
</tr>
<tr>
<td>---------------------</td>
<td>--------</td>
<td>--------------</td>
<td>--------------------</td>
<td>-----------------------</td>
<td>------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cleat angle</td>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Top Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td>Fully welded BCF</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td>Fully welded BCW</td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td></td>
<td>I</td>
</tr>
<tr>
<td>Flanges welded BCF</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>I</td>
</tr>
<tr>
<td>Flanges welded BCW</td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td></td>
<td>I</td>
</tr>
<tr>
<td>End plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
</tr>
<tr>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT&quot;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td></td>
<td>I</td>
<td></td>
</tr>
<tr>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>NO</td>
<td>HSS_RECT, HSS_CIRC</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fin plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Column &quot;YES&quot;</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT, HSS_CIRC&quot;</td>
</tr>
<tr>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>Beam &quot;I&quot; - Column &quot;I, HSS_RECT, HSS_CIRC&quot;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td></td>
<td>I</td>
<td></td>
</tr>
<tr>
<td>Flange cover plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td></td>
<td>I</td>
</tr>
</tbody>
</table>
### Table 52: IS 800-2007

<table>
<thead>
<tr>
<th>Connection</th>
<th>Family</th>
<th>Inclination angles</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Skew</td>
</tr>
<tr>
<td><strong>Cleat angle</strong></td>
<td>BCF</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>No</td>
</tr>
<tr>
<td><strong>Web side plate</strong></td>
<td>BCF</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Yes</td>
</tr>
<tr>
<td><strong>Moment end plate</strong></td>
<td>BCF</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>BS</td>
<td>No</td>
</tr>
<tr>
<td><strong>Bolted splice web plates</strong></td>
<td>BS</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>CS</td>
<td>No</td>
</tr>
<tr>
<td><strong>Flexible end plate</strong></td>
<td>BCF</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Yes</td>
</tr>
<tr>
<td>Connection</td>
<td>Family</td>
<td>Inclination angles</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>---------</td>
<td>--------------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Skew</td>
</tr>
<tr>
<td>Flange angles</td>
<td>BCF</td>
<td>No</td>
</tr>
<tr>
<td>Seating angles</td>
<td>BCF</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>No</td>
</tr>
<tr>
<td>Bolted splice flange plates</td>
<td>BS</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>CS</td>
<td>No</td>
</tr>
<tr>
<td>Base plate: Column</td>
<td>BP</td>
<td>No</td>
</tr>
</tbody>
</table>

Table 53: AS 4100-1998

<table>
<thead>
<tr>
<th>Connection</th>
<th>Family</th>
<th>Member Types</th>
<th>Built-Up Symmetric</th>
<th>Built-Up Asymmetric</th>
<th>Member Flange Rotation</th>
<th>Sections Allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base plate</td>
<td>CB</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I, I2C, HSS_RECT, HSS_CIRC (Column)</td>
</tr>
<tr>
<td>Bolted end plate</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BS</td>
<td>Haunched</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I, T</td>
</tr>
<tr>
<td></td>
<td>BCF</td>
<td>Prismatic (Column, Beam)</td>
<td>YES</td>
<td>YES</td>
<td>YES (Column)</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCF</td>
<td>Haunched (Beam)</td>
<td>YES</td>
<td>YES</td>
<td>YES (Column)</td>
<td>I, T (Haunch)</td>
</tr>
<tr>
<td>Mitred Knee</td>
<td>BCF</td>
<td>Prismatic</td>
<td>Yes</td>
<td>Yes</td>
<td>YES (Column)</td>
<td>I</td>
</tr>
<tr>
<td>Angle cleat</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Column)</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Flange plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Column)</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Connection</td>
<td>Family</td>
<td>Member Types</td>
<td>Built-Up Symmetric</td>
<td>Built-Up Asymmetric</td>
<td>Member Flange Rotation</td>
<td>Sections Allowed</td>
</tr>
<tr>
<td>-----------------------------</td>
<td>--------</td>
<td>--------------</td>
<td>---------------------</td>
<td>----------------------</td>
<td>-------------------------</td>
<td>------------------------</td>
</tr>
<tr>
<td>Seating connections</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Column)</td>
<td>I, HSSRECT (Column)</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I, HSSRECT (Column)</td>
</tr>
<tr>
<td>End plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Column)</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Web side plate</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Column)</td>
<td>I (Beam)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>I, HSSRECT, HSSCIRC (Column)</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I (Beam)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>I, HSSRECT, HSSCIRC (Column)</td>
</tr>
<tr>
<td></td>
<td>BG</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td>Bolted flange plates</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Top Column)</td>
<td>I</td>
</tr>
<tr>
<td>Bolted web plates</td>
<td>BS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>CS</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Top Column)</td>
<td>I</td>
</tr>
<tr>
<td>Bearing pad</td>
<td>BCF</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Top Column)</td>
<td>I (Beam)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>I, HSSRECT, HSSCIRC (Column)</td>
</tr>
<tr>
<td></td>
<td>BCW</td>
<td>Prismatic</td>
<td>YES</td>
<td>YES</td>
<td>YES (Top Column)</td>
<td>I (Beam)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>I, HSSRECT, HSSCIRC (Column)</td>
</tr>
</tbody>
</table>
D. Using the RAM Connection mode

This section provides you with some common procedures used in design steel connections in the Connection Design mode.

D. To edit the RAM Connection settings

1. Click the RAM Settings tool. The RAM Connection Settings dialog opens.
2. Select the default Design Code from the drop down list.
3. (Optional) For AISC codes, select the Consider AISC 341-05 and AISC 358-05 Seismic Provision option to include these seismic checks in design of connections by default for AISC codes.
4. (Optional) Select the Design multiple selected connections individually option to disable grouping of connections into same templates.
5. Check the country names to include various section catalogs in the Sections Add to RAM list.
6. Click OK to save the changes.

Related Links
- D. RAM Connection Settings dialog (on page 977)
- D. RAM Connection Settings dialog (on page 977)

D. To design steel connections

Before performing a connection design, you should select the appropriate design code and options in the RAM Connection Settings dialog.

1. Run an analysis.
Tip: The Connection Design workflow is only available after the completion of a successful analysis.

2. Select **Connection Design** in the Workflows panel.

The **RAM Connection Validation** dialog opens.

3. Click **Close**.

The **RAM Connection Input** dialog opens.

4. Select the **Add Load Envelope** tool.

The **Design Load Envelope** dialog (on page 976) opens.

5. Select the loads you wish to use for design and then click **OK**.

6. Select the joint(s) you wish to add connections to.

   **Note:** Use the selection tools on the **Connection Design** ribbon tab in the **Assign Connections** group to assist selecting the intended joints.

   Selected joints are highlighted with red dots.

7. In the **Connection Design** ribbon tab in the **Assign Connections** group, select the appropriate tool corresponding to the connection assignment you want to make:

   - **Basic Connection** (on page 967)
     
   or

   - **Smart Connection** (on page 968)
     
   or

   - **Gusset Connection** (on page 968)
     
     In the corresponding Connection dialog, you can select which design code and the connection template you wish to use for the selected joint's design brief.

8. Click **OK** to assign the templates and design the connections.

   The Connection Design dialog will display a list of all connection assignments.

9. Click **Close** to dismiss the dialog.

   A list of connections and basic information is displayed in the **RAM Connection Input** table (on page 974).

D. To select a basic connection template

An appropriate set of joints must be selected in order to assign basic connections.

1. Either:

   - on the **Connection Design** ribbon tab, select the **Basic Connection** tool in the **Assign Connection** group

   or
in the **RAM Connection Input** dialog, select the **Basic Connections** tool. The **Basic Connection** dialog opens.

2. You may specify design code or grouping overrides here.

3. Select a connection class from the drop-down list.
   All available connections are displayed in the **Available** list box below.

4. Select one or more connections and add them to the **Selected** list box by clicking the [>] (add) button.

   **Tip:** You may add all connections in the list by clicking the [>>] (add all) button.

5. Click **OK** to accept and design the selected joints using these connections.

**Related Links**

- **Basic Connections dialog** (on page 3166)

**D. To Select a Smart Connection Template**

An appropriate set of joints must be selected in order to assign smart connections.

1. Either:

   Click the **Smart Connections** tool

   or

   Select **Connection Design > Assign Smart Connection**
   The **Smart Connection** dialog (on page 3169) opens.

2. You may specify design code or grouping overrides here.

3. (Optional) For AISC codes, select the option for **Consider AISC 341-05 Seismic Provisions**
   Select a load cases from the drop-down list which represents the gravity load.

4. (Optional) For BS5950 code, select the option to **Consider Structural Integrity** if necessary.

5. (Optional) For Base Plates, select a **Seismic Category** site class from the drop down list.

6. Select a connection class from the drop-down list.
   All available connections are displayed in the **Available** list box below.

7. Select the connection you wish to use and add it to the Selected list box by clicking the Add (>) button.

8. Click **OK** to accept and design the selected joints using these connections.

**Related Links**

- **Smart Connections** (on page 3169)

**D. To Select a Gusset Connection Template**

An appropriate set of joints must be selected in order to assign gusset connections.

1. Either:

   Click the **Gusset Connections** tool

   or
Select **Connection Design** > **Assign Gusset Connection**

The [Gusset Connection dialog](on page 3171) opens.

2. You may specify design code or grouping overrides here.

3. Select a connection class from the drop-down list. All available connections are displayed in the **Available** list box below.

4. Select the connection you wish to use and add it to the Selected list box by clicking the Add (>) button.

5. Click **OK** to accept and design the selected joints using these connections.

**Related Links**

- [Gusset Connections dialog](on page 3171)

**D. To design an HBBB connection**

To design a horizontal brace-beam-beam gusset connection, use the following procedure.

**HBBB connections apply to joints with the following conditions:**

- no column at the joint
- two perpendicular beams with H section and one brace with an angle section
- beams should be parallel with the ground (X-Z plane when Y is up)
- the brace should be in the same plane with the beams, with the angle between the brace and either beam between 30° and 60°

1. Start the **Connection Design** workflow and select the applicable load envelope.

2. Select a joint that matches the joint requirements.

3. On the **Connection Design** ribbon tab, select the **Gusset Connection** tool in the **Assign Connections** group.

The **Gusset Connection** dialog opens.

4. Select an AISC design code from the **Design Code** drop-down list.

5. Select **Gusset Plate HBBB** from the connection template drop-down list.

6. Double-click the **HBBB_CA** template to add it to the selected list.

7. Click **OK**.

8. Double-click the connection list view to review the connection design. The connection pad opens.

9. (Optional) If the girder and beam have been incorrectly specified, they can be changed:
   - a. On the **Connection Design** ribbon tab, select the **Beam and Girder Identification** tool in the **Frames** group.
   - The **Beam-Girder Identification** dialog opens.
   - b. Check the Swap option for the beam-girder connection you want to switch.
   - c. Click **Switch**.
   - d. Click **OK**.
   - e. Repeat step 8 to open the connection design again.

**D. To design an HCBB connection**

To design a horizontal brace-column-beam gusset connection, use the following procedure.

**HCBB connections apply to joints with the following conditions:**
• at least one column with an H section at the joint
• column is vertical (perpendicular to beams)
• two perpendicular beams with H section and one brace with an angle section
• beams should be parallel with the ground (X-Z plane when Y is up)
• the brace should be in the same plane with the beams, with the angle between the brace and either beam between 30° and 60°

1. Start the **Connection Design** workflow and select the applicable load envelope.
2. Select a joint that matches the joint requirements.
3. On the **Connection Design** ribbon tab, select the **Gusset Connection** tool in the **Assign Connections** group.

   The **Gusset Connection** dialog opens.

4. Select an AISC design code from the **Design Code** drop-down list.
5. Select **Gusset Plate HCBB** from the connection template drop-down list.
6. Double-click the **HCBB_CA** template to add it to the selected list.
7. Click **OK**.
8. Double-click the connection list view to review the connection design.

D. To edit steel connections

1. Either:

   On the **Connection Design** ribbon tab, select the **Joint Cursor** tool in the **Assign Connections** group and then double-click on any connection in the View window.

   or

   Select the entry for the connection row in the **RAM Connection Input** table and then click **Edit**

   The **Connection Pad** opens displaying the design information for this joint.

2. Make the desired changes to the connection input data.
3. Select the **Save** tool to save any changes.

Changes made to the connection pad are saved back to the connection design in STAAD.Pro.
4. Click the [X] to close the Connection Pad window.

D. Selecting Joints & Connections

The Select Nodes tool is used to select nodes in the View window for assigning connection templates.

The Select Joints tool (found in the Selection toolbar or from the Select menu) is used to select connections in the View window. Typical STAAD.Pro graphical selection rules apply.

The Select menu contains tools to select different connections and joints based on logical criteria:

- The Select > Select Joints sub-menu contains tools which logically select joints based on the connecting members.
- The Select > Select Connections sub-menu contains tools which aide in selecting joints for grouping.

D. Design Connections Individually

Any time multiple connections fitting the same template are selected and a template is applied, these will be grouped to the template meeting the minimum requirement of the entire group.

If you wish to design each connection to an optimized (and likely different) template, then select the Design Connections Individually option in the Connection assignment dialog before clicking OK.

This option can be selected in the RAM Connection Settings dialog (on page 977) to disable the grouping feature.

Related Links

- Basic Connections dialog (on page 3166)
- Gusset Connections dialog (on page 3171)
- Smart Connections (on page 3169)

D. To delete steel connections

1. Select the connection(s) you wish to remove in the RAM Connection Input table.
2. Either:
   - press <Delete>
   - or
   - click Delete
3. Click Yes to confirm the deletion.

Tip: You may need to Refresh the display (press <F5>) to see the connection icon removed in the View window.

D. To Export Connection Designs to a Report

To export the connection design results to a Microsoft® Office Word® document report, use the following procedure.

You must perform connection design on one or more connections to generate a report.

1. Either:
On the **RAM Connection Input** Table, select the **Export Connection Reports** tool

or

Select **Connection Design > Export Connections report**.

The **RAM Report Export** dialog opens.

2. Select the connections you want to include in the report.

   **Tip:** Click **Select All** to quickly include all connection design into the report.

3. (Optional) Select or clear options to include **Data Report** Result Report forumla

4. Select to output multiple connections to either **Individual Reports** (separate file for each connection) or a **Merged Report** (single file containing all selected connections).

5. Click **OK**.

The report is generated and saved in the same folder as the STAAD.Pro input file.

**Related Links**
- **RAM Report Export dialog** (on page 3175)

**D. To add connection designs to your report**

To add a summary of steel connection design results to your STAAD.Pro report, use the following procedure.

1. Select the **Postprocessing** workflow.

2. Either:
   - select the **Reports** page
   - select the **File** ribbon tab and then select **Report > Setup** in the backstage view

      The **Report Setup** dialog opens.

3. Select the **Items** tab.

4. In the **Available** topics drop-down list, select **RAM Connection Summary Report**.

5. Double-click the **RCNX Report** entry to add it to the **Selected** list.

      The connection design summary is now included in the report.

6. Either:
   - click **Print** to generate a copy of the report
   - click **OK** to dismiss the dialog and review the report on screen.

**D. Custom Connection Templates**

You can create and use your own connection templates.

**D. To create a custom template file**

Custom template files are created in the **Connection Design** workflow.

1. On the **Connection Design** ribbon tab, select the **Connection Database** tool in the **Configure** group.

      The connection database dialog opens.
2. Click **New group**.
3. Type a New group name then click the X to close the dialog.
   The new group is created and selected as the current group.
4. Click **New table**.
   The new table dialog opens.
5. Specify the table parameters:
   a. Select the **Design code** to use for these connections.
   b. Select the **Connector type** from the drop-down list.
   c. Select the **Joint type** for the connections.
   d. Select the **Connector name** for the connection.
   e. The context-sensitive help panel provides additional information on the connector name selection.
   f. Click **OK**.

   ![New connection dialog](image)

   The new table is saved to the custom group.
6. (Optional) Repeat Steps 4 and 5 to create additional tables as needed.

D. To add a connection template to a table
To add a custom connection template to a database table, use the following procedure.
Tip: One of the simplest methods to get started is to copy factory installed templates to your new group and edit those to suite your needs.

1. On the Connection Design ribbon tab, select the Connection Database tool in the Configure group. The connection database dialog opens.
2. Select Custom in the Groups drop-down list.
3. Click New Item. The Connection Pad window opens.
4. Specify the connection parameters:
   a. Type the connection Name.
   b. Update the connection parameters for this connection type.
   c. Click Save.
   d. Close the window.
   The new connection name is added to the items list. A list of the connection tables where you can add this connection type are also listed.
5. Click >> to add the new connection to the connection database file.

The new connection type is available for use in design in this connection database.

**Note:** When the connection database is selected in the connection database dialog, you can click Reset to remove any custom items and restore the database to the default values.

D. Pages in the Connection Design workflow

The page control bar in Connection Design workflow contains the following pages:

**D. Connections page**

Used to select, assign, design, and edit steel connections. The Connection page is where connections are defined and designed.

When the Connections page is selected, the RAM Connection Input table opens.

The Whole Structure view window displays the structure (or a portion thereof), which is used to graphically select joints for connection assignment and connections for editing.

**Tip:** When a table row(s) is selected in the RAM Connection Input table, the corresponding Joint(s) will be highlighted in the view window.

**D. RAM Connection Input table**

Contains tools to generate new connection designs and contains all previously designed connections.

Opens when the Connection | Connection page (on page 974) is selected.
Tools

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
<th>Same Effect as Selecting</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Icon" /></td>
<td><strong>Load Envelope</strong></td>
<td>Opens the [Design Load Envelope dialog](on page 976), which is used to select load cases and combinations to include in the load envelope for steel connection design.</td>
</tr>
<tr>
<td><img src="image2.png" alt="Icon" /></td>
<td><strong>RAM Connection Settings</strong></td>
<td>Opens the [RAM Connection Settings dialog](on page 977), which is used to set the default design code and grouping toggle.</td>
</tr>
<tr>
<td><img src="image3.png" alt="Icon" /></td>
<td><strong>Assign Basic Connections</strong></td>
<td>Opens the [Basic Connections dialog](on page 3166), which is used to assign basic connection templates to selected joints.</td>
</tr>
<tr>
<td><img src="image4.png" alt="Icon" /></td>
<td><strong>Assign Smart Connections</strong></td>
<td>Opens the [Smart Connections dialog](on page 3169), which is used to assign smart connection templates to selected joints.</td>
</tr>
<tr>
<td><img src="image5.png" alt="Icon" /></td>
<td><strong>Assign Gusset Connections</strong></td>
<td>Opens the [Gusset Connections dialog](on page 3171), which is used to assign gusset connection templates to selected joints.</td>
</tr>
</tbody>
</table>
### Design

#### D. Steel Design

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
<th>Same Effect as Selecting</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="icon.png" alt="Icon" /></td>
<td><strong>Show RAM Database</strong>&lt;br&gt;Opens the [RAM Connection Material Database dialog](on page 3176), which is used to review additional material data required by RAM Connection for the design of steel connections.</td>
<td><strong>Connection Design &gt; Assign RAM Materials</strong></td>
</tr>
<tr>
<td><img src="icon.png" alt="Icon" /></td>
<td><strong>Identify Beam-Girder</strong>&lt;br&gt;Opens the [Beam-Girder Identification dialog](on page 3174), which is used to switch beam and girder assignments in a Beam-Girder connection.</td>
<td><strong>Connection Design &gt; Identify Beam and Girder</strong></td>
</tr>
<tr>
<td><img src="icon.png" alt="Icon" /></td>
<td><strong>Export Connection Reports</strong>&lt;br&gt;Opens [RAM Report Export dialog](on page 3175), which is used to manage connections and other report details to include in an report.</td>
<td><strong>Connection Design &gt; Export Connections report</strong></td>
</tr>
</tbody>
</table>

**Table**

Once a Connection is established the following information will be displayed in the Input grid:

- Connection Name
- Template used for the Connection
- Description of the Connector
- Node (where the joint is created)
- Members which participated in the connection

**D. Design Load Envelope** dialog

Used to specify the loads which will be included in the Load Envelope used for steel connection design.

Opens when **Connection Design > Load Envelope for Connection** is selected.
Defined Envelopes
Select this option to use a load envelope which has been previously defined for this model. A list of all defined envelopes is then available in the drop-down list. The option will be inactive if no load envelopes have been defined.

Defined Loads
This option allows you to select loads or load combinations to be included in the connection design load envelope. The following tools are used to specify which loads should be included in the envelope.

- **Select All Load Cases Show Below** — Selecting this will select all loads displayed in the load list box.
- **Show Combinations Only** — Selecting this option will limit the load list to load combinations.
- **Load List** — A list of all loads in the model are displayed here. Each load in the list has an associated box to select it for inclusion in the load envelope.

**OK**
Click this button to accept changes made to the Design Load envelope.

**Cancel**
Click this button to close the dialog with no changes saved.

**Help**
This button opens the online documentation.

### D. RAM Connection Settings dialog
Used to set the default design code and grouping toggle.

Opens when the **RAM Settings** tool is selected in the RAM Connection input dialog.
Specify the default design code to be used from the drop-down list. The following codes are currently available in Connection Design in STAAD.Pro:

- AISC 360-10 (ASD) — Allowable Stress Design per AISC 360-10
- AISC 360-10 (LRFD) — Load Resistance Factor Design per AISC 360-10
- AISC 360-05 (ASD) — Allowable Stress Design per AISC 360-05
- AISC 360-05 (LRFD) — Load Resistance Factor Design per AISC 360-05
- AS 4100 1998
- BS 5950-01:2000
- EN 1993-1-7 — Eurocode 3
- IS 800-2007
- GB50017-2003 — "Design of Steel Structures" Chinese code

Consider AISC 341-05 and AISC 358-05 Seismic Provision

(AISC codes only) Select this option to direct the program to consider seismic provisions per AISC 341-05 “Seismic Provisions for Structural Steel Buildings” and AISC 358-05 “Prequalified Connections for Special and Intermediate Steel Moment Frames for Seismic Applications” in the design of connections of members in a designated seismic frame.

Design multiple selected connections individually

Select this option to disable grouping of connections into same templates.

Sections Add to RAM

Check the boxes associated with country section catalog to include in the RAM Connection catalog.

OK

Accepts the settings changes and closes the dialog.

Cancel

Closes the dialog without saving changes.

Help

Opens the Help window.

Related Links

- D. To edit the RAM Connection settings (on page 966)
- D. To edit the RAM Connection settings (on page 966)

D. Results page

Used to review steel connection designs. The Results page displays the summary, layout, and results of the connection designs.

View Window

When a row of a RAM Connection Result table is selected, the DXF drawing and the Report will be displayed in the View window.

RAM Connection Results table

Contains the following items for each connection (upon successful connection design):

- Joint Name
- Template
Design
D. Steel Design

- Design Code
- Ratio (Critical Strength Ratio)
- Status — Possible values:
  - OK (Critical Strength Value < 1),
  - No Good (Critical Strength Value > 1), or
  - Warning (This connection may require additional review)

D. Seismic Frames page

Used to assign and review seismic frames for connection design.

When the Seismic Frame page is selected in the Connection workflow, the RAM Connection Result frames table opens.

D. RAM Connection Result frames table

Used to review frame member assignments and specify reduced section (induced plastic hinge) parameters for beam members within a frame.

Note: A column member may be used in multiple frames but a beam member can only be used in a single frame definition.

Opens when the Seismic Frame page (on page 979) is selected in the Connection workflow.

![RAM Connection Result frames table]

The table rows are grouped first by Frame No and then by Member number. The Type of lateral seismic resisting is also displayed. These elements (shaded in yellow) may not be edited. The location and dimensions of reduced beam sections (RBS) for forming plastic hinges may be specified here for beam members.

Where:

- \( Sh \) is the distance to the Hinge Location
- \( RBS-a \) is the horizontal distance to locate the reduce beam section
- \( RBS-b \) is the length of the reduced section of the flange of the beam
- \( RBS-c \) is the depth of the reduced section of the flange

D. Application Window Layout

After entering the Connection Design Mode is entered, some of the menus change to offer some additional tools.

Note: For help with all other application window elements, please refer to the Modeling mode section of the User Interface help (on page 639).
D. RAM Connection pad

Used to inspect connection details, edit a connections input, and access steel connection design results. Opens when a connection icon is double clicked in the View window.

**Note:** The appearance and layout of the Connection pad varies depending on which version of RAM Connection is installed. The tools and operation remain essentially the same.

The connection pad is required for the creation of templates for the database and also to review/edit a model connection, or many model connections with the connections detailer. It is accessed when any template of the database is edited, when the user double clicks a model connection, or when several identical model connections are selected and the detailing command is invoked from the main menu.

D. Save Graphic dialog

Used to save a connection drawing to an external image file.

Opens when the **Print to File** tool is selected in the **Connection Pad Drawing** window.
**File name**  
Specify a name for the image file.

**Browse**  
Opens a Windows Save As dialog, which is used to specify a drive, directory, and file name for the image file.

**Format**  
Select one of the following common image file formats:

- **Windows Bitmap (BMP)** - An uncompressed image file format. These are very common and readable by most image editing tools. Many word processing or spreadsheet programs can import these image files for re-use. These files can be very large for even relatively short reports (approximately 20x the size of a JPEG).

- **JPEG graphic (JPG)** - A compressed image format commonly used for photographic images. These are also readable by a wide variety of programs, including web browsers, image editors, and office programs. JPEG is not recommended for line drawings or text due to the blurring that may result from its compression algorithm.

**Scale**  
Specify a scale factor to be used with the image file.

**Width**  
Specify a width (in pixels) for the image file.

**Height**  
Specify a height (in pixels) for the image file.

**Block aspect ratio**  
Selecting this option will scale the height or width such that the aspect ratio of the original drawing is constant in the image file.

**OK**  
Create the image file with the selected options and close the Save Graphic dialog.

**Cancel**  
Close the dialog without saving an image file.
D. Connection Tags

Connection tags are used to associate connection data included in an XML file with members. This data can be exchanged with other programs via the ISM link. The connections can also be checked by defined capacities in the XML file.

Connection tags consist of two pieces of data:

i. A Connection Tags XML file, which contains the connection categories, tag names, and member end releases for the connection tag. Connection capacities are also specified for each combination of member and connecting member which may utilize a connection tag. Refer to Connection Tags XML File Schema (on page 986) for additional information on the required structure of this XML file.

ii. Assignments of connection tags to members are stored in the STAAD input file. Though this is done within the DEFINE MEMBER ATTRIBUTE command, it is strongly recommended that the user interface features be used to make connection tag assignments as these must utilize only the connection categories and tag names in the associated XML file. Refer to TR.29.2 Connection Tag Member Attribute (on page 2537) for additional information on this command.

D. To create a connection tag

To create a new connection tag for use in the current STAAD model, use the following steps.

Note: You must have a Connection Tags XML file containing appropriate connection capacities with members used in the model. A default XML file is included with the program that contains a minimal amount of connection data.

A connection tag is added to the model to define the type of connection (e.g., shear or moment) and the associate attributes of that connection (e.g., end releases or capacities associated with the connection).

1. Select one or more members in the view window.
2. Either:
   - on the Beam Tools ribbon tab, select the Assign tool in the Connection Tags group

   or

   - right-click and select Connection Tags > Assign Connection Tags from the pop-up menu

   Note: If you have not specified a Connection Tags XML data file, then the Select Connection Tag File dialog opens prompting you to select an appropriate XML file.

   The Assign Connection Tags dialog and New Connection Tag dialog open.

3. If the New Connection Tag dialog did not already open, then click the New tool in the dialog toolbar.
4. On the New Connection Tag dialog, select the category to use from the Select Categories drop-down list.
5. Select the Tag to use from the selected category from the Select Tags drop-down list.
6. Either:
   - set the option to Assign Beam End Releases to the use the end releases in the connection tag definition
or clear the Assign Beam End Release option and manually select the end releases you want for this connection.

7. Select either the Start or End Location option (or both).

**Note:** Selecting both Start and End locations creates two distinct connections, which must both be assigned.

8. Either:
   - click Add to add this connection to the connection tags list in the Assign Connection Tags dialog
   - or
   - click Assign to assign the connection tag to the current member selection

**Related Links**
- Assign Connection Tags dialog (on page 3183)
- New Connection Tag dialog (on page 3185)

### D. To delete a connection tag

To remove a connection tag entirely from the STAAD model, use the following steps.

This procedure will remove the connection from the model. To remove the assignment of a connection tag from one or more members, see D. To remove connection tag assignments (on page 984).

1. Select one or more members in the view window.
2. Either:
   - on the Beam Tools ribbon tab, select the View tool in the Connection Tags group
   - or
   - on the Utilities ribbon tab, select the Connection Tags > View Tags tool in the Tools group
   - or
   - right-click and select Connection Tags > View Connection Tags from the pop-up menu.
   - The Assign Connection Tags dialog opens.
3. Select the row in the Connection Tags table for the connection tag you want to remove.
4. Select the Remove tool in the dialog toolbar.
   - A confirmation dialog opens to confirm you want to delete the connection tag.
5. Click OK.
   - A confirmation dialog opens in the STAAD.Pro user interface window to confirm you want remove the associated STAAD input command for the member attribute definition.
6. Click Yes.
   - The row is removed from the Connection Tags table.
D. To remove connection tag assignments

To remove the connection tag assignments from members, use the following steps.

This procedure will disassociate a connection tag with a member end. If you want to complete remove a connection tag from the STAAD.Pro model, see D. To delete a connection tag (on page 983).

1. Select one or more members in the view window.
2. Either:
   - on the **Beam Tools** ribbon tab, select the **Remove** tool in the **Connection Tags** group
   - right-click and select **Connection Tags** > **Remove Connection Tags** from the pop-up menu

The **Assign Connection Tags** and the **Remove Connection Tag** dialogs open.

3. Select either the **Start** or **End** (or both) corresponding to the member end from which you want the connection tag removed.
4. (Optional) If you do not want the beam end releases for this beam end changed, then clear the option **Remove Beam End Releases**.

   **Tip:** By default, this option is set so the member end releases for this end are returned to their defaults (i.e., fully fixed).

5. Click **Remove**.
   A confirmation dialog opens.
6. Click **OK**.
   A confirmation dialog opens in the STAAD.Pro user interface window to confirm you want remove the associated STAAD input command for the member attribute definition.
7. Click **Yes**.
   The member number is removed from the Assigned Beams cell of the Connection Tags table.

   **Note:** If the tag selected was only assigned to a single member end, then the row is removed from the Connection Tags table.

**Related Links**

- **Remove Connection Tags dialog** (on page 3187)
D. To check connection tags

To check the assigned connection tags capacity using the STAAD.Pro analysis results, use the following steps.

Prior to checking connections, you must perform a successful analysis on the model so analysis results at member ends can be compared to connection capacities.

Additionally, each beam and connecting member (beam or column) must have a connection capacity for the sections used in the associated Connection Tags XML file to facilitate checking connections.

1. Select one or more members in the view window.
2. Either:
   - on the Beam Tools ribbon tab, select the Check tool in the Connection Tags group
   - or
   - on the Utilities ribbon tab, select the Connection Tags > Check Tags in the Tools group
   - or
   - right-click and select Connection Tags > Check Connection Tags from the pop-up menu
   - The Assign Connection Tags dialog and Check Connection Tags dialog open.
3. Select the load case types you want to use in the Select Load Case Types drop-down list. The load cases list displays only the type or types selected.
4. Either:
   - select load cases individually by setting their check boxes on in the list
   - or
   - set the Select All option check box on.
5. Click Check.
   - The Connection Tag Check Results table opens. This table displays the connection tag data from the Connection Tag XML file capacity used checked against the critical load case at each member end with a connection tag. The status of the check (i.e., Pass or Fail) is listed in the Remarks column.

Related Links
- D. Connection Tags Capacity Checks (on page 985)
- Check Connection Tags dialog (on page 3100)

D. Connection Tags Capacity Checks

The rules for checking the connections are dependent on the whether the connection is typed as moment or shear.

**Moment Connections**

Shear Capacity Check - The shear capacity for a given beam/column combination present in the ConnectionTagFile.xml is checked against the demand taken as the absolute value of the major axis shear value, Fy, from each of the load cases included and the maximum value reported.
If the shear capacity in the XML file is 207.345 kN, then:

- when \( F_y = \pm 208.0 \text{ kN} > 207.345 \text{ kN} \), the connection check is failed.
- when \( F_y = \pm 200.0 \text{ kN} < 207.345 \text{ kN} \), the connection check is passed.

**Moment Capacity Check** - The moment capacity check for a given beam/column combination is based on both the major axis moment and axial force. For each load case, the maximum demand is taken from the end moment plus the axial force multiplied by a specified alpha factor and compared against the given moment capacity.

If the moment capacity in the XML file is 35.951 kN·m and the tension force factor, alpha, is 0.09565, then

\[
\text{demand} = |M_z| + \alpha \cdot |F_x|
\]

- when \( M_z = 30 \text{ kN·m} \) and \( F_x = -50 \text{ kN} \), then demand is \( 30 + 0.09565(50) = 34.7825 \text{ kN·m} < 35.951 \text{ kN·mm} \), the connection check is passed.
- when \( M_z = 32 \text{ kN·m} \) and \( F_x = -50 \text{ kN} \), then demand is \( 32 + 0.09565(50) = 36.7825 \text{ kN·m} > 35.951 \text{ kN·mm} \), the connection check is failed.

**Shear Connections**

The shear capacity for a given beam/column combination present in the `ConnectionTagFile.xml` is checked against the demand taken as the major axis shear value, \( F_y \), and a proportion of the axial force, \( F_x \), from each of the load cases included and the maximum value reported.

If the shear capacity in the XML file is 15.538 kN and the tension force factor, alpha, is 0.2016, then demand = \( |F_y| + \alpha \cdot |F_x|\):

- when \( F_y = 12 \text{ kN} \) and \( F_x = -25 \text{ kN} \), then demand = \( 12 + 0.2016(25) = 17.04 \text{ kN} > 15.538 \text{ kN} \), the connection is failed.
- when \( F_y = 10 \text{ kN} \) and \( F_x = 25 \text{ kN} \), then demand = \( 10 + 0.2016(25) = 15.04 \text{ kN} < 15.538 \text{ kN} \), the connection is passed.

Note: There are no moment capacity checks for a shear connection.

**Related Links**

- [D. To check connection tags](on page 985)
- [Check Connection Tags dialog](on page 3100)

**D. Connection Tags XML File Schema**

A simple schema is used to define the data for a Connection Tags XML file used for storing connection tag data for STAAD.Pro.

When you initiate a connection tags action for the first time from the right-click pop-up menu, you will be prompted to select a Connection Tags XML file for use with creating and checking connection tags.
ConnectionTagFile

The `<ConnectionTagFile>` element is the root element for a STAAD.Pro Connection Tags file. Within a ConnectionTagFile element two types of data: connection categories and connection tags.

Table 54: ConnectionTagFile model details

<table>
<thead>
<tr>
<th>Contains</th>
<th>(File Version then Categories then Equations then Tags)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>n/a (root element)</td>
</tr>
<tr>
<td>Attributes</td>
<td></td>
</tr>
</tbody>
</table>

FileVersion

The `<FileVersion>` element is indicate the file version used.

Table 55: FileValue model details

<table>
<thead>
<tr>
<th>Contains</th>
<th>n/a (empty element)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>ConnectionTagFile</td>
</tr>
<tr>
<td>Attributes</td>
<td>value (required, decimal)</td>
</tr>
</tbody>
</table>

Categories

The `<Categories>` element contains individual connection categories. Typically a connection tags file contains both Moment and Shear categories.

Table 56: Categories model details

<table>
<thead>
<tr>
<th>Contains</th>
<th>(Category) (one or more)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>ConnectionTagFile</td>
</tr>
<tr>
<td>Attributes</td>
<td></td>
</tr>
</tbody>
</table>

Category

The `<Category>` element defines a connection category.

Table 57: Category model details

<table>
<thead>
<tr>
<th>Contains</th>
<th>(CategoryDesc)</th>
</tr>
</thead>
</table>
**CategoryDesc**

The `<CategoryDesc>` element is used to provide a human readable description for the parent connection category. It can contain any alpha-numeric string.

**Table 58: CategoryDesc model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>text data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Category</td>
</tr>
<tr>
<td>Attributes</td>
<td></td>
</tr>
</tbody>
</table>

**Equations**

The `<Equations>` element contains individual equations.

**Table 59: Categories model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>(Equation) (one or more)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>ConnectionTagFile</td>
</tr>
<tr>
<td>Attributes</td>
<td></td>
</tr>
</tbody>
</table>

**Equation**

The `<Equation>` element contains a user-defined equation.

**Table 60: Capacity model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>n/a (empty element)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Equations</td>
</tr>
</tbody>
</table>
Attributes

<table>
<thead>
<tr>
<th>Attributes</th>
<th>EquationID (required, string), Equation (required, string), Condition (required, string), Limit(required, decimal)</th>
</tr>
</thead>
</table>

See “Equations Guidelines” (on page 993) for details on writing equation and condition values.

Tags

The `<Tags>` element contains the connection tag data in a series of child elements.

**Table 61: Tags model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>(Tag) (one or more)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>ConnectionTagFile</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Attributes</th>
<th></th>
</tr>
</thead>
</table>

Tag

The `<Tag>` element contains an end release definition and a set of connection capacities to be used for checking connection tags.

**Table 62: Tag model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>(EndRelease then Capacities)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Tags</td>
</tr>
<tr>
<td>Attributes</td>
<td>TagName (required, string, no default), CategoryName (required, restricted string data, no default); values are: MOMENT SHEAR</td>
</tr>
</tbody>
</table>

EndRelease

The `<Tag>` element is used to specify the end release conditions to be used with a connection tag.

**Table 63: EndRelease model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>n/a (empty element)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Tag</td>
</tr>
</tbody>
</table>
Attributes

<table>
<thead>
<tr>
<th>Attributes</th>
<th>FX (required, restricted integer), FY (required, restricted integer), FZ (required, restricted integer), MX (required, restricted integer), MY (required, restricted integer), MZ (required, restricted integer)</th>
</tr>
</thead>
</table>
|            | The range for these attributes is:  
|            | 0 - restrained  
|            | 1 - full release |

**Capacities**

The `<Capacities>` element contains one or more connection capacity value sets for a specific system of units.

**Table 64: Capacities model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>(Capacity) <em>(one or more)</em></th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Tag</td>
</tr>
</tbody>
</table>
| Attributes | UnitSystem(required, restricted string data, no default); values are:  
|            | METRIC  
|            | IMPERIAL |

**Beam**

The `<Beam>` element contains one or more beam or column sections.

**Table 65: Beam model details**

<table>
<thead>
<tr>
<th>Contains</th>
<th>BeamOrCol <em>(one or more)</em></th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Capacities</td>
</tr>
<tr>
<td>Attributes</td>
<td>Name (required, string)</td>
</tr>
</tbody>
</table>

**BeamOrCol**

The `<BeamOrCol>` element is used to specify connection capacity bending and shear capacities for a specific member, or for the default member.
Table 66: BeamOrCol model details

<table>
<thead>
<tr>
<th>Contains</th>
<th>n/a (empty element)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Beam</td>
</tr>
<tr>
<td>Attributes</td>
<td>Name (required, string), Mz.cap (required, decimal), Fx.cap (required, decimal), Fz.cap (required, decimal), alpha (required, decimal)</td>
</tr>
<tr>
<td></td>
<td>See D. Connection Tags Capacity Checks (on page 985) for details on how these values are used.</td>
</tr>
<tr>
<td></td>
<td>See “Use of Wild Cards (on page 993)” for details on using defaults.</td>
</tr>
</tbody>
</table>

**Checks**

The `<Checks>` element contains one or more check.

Table 67: Checks model details

<table>
<thead>
<tr>
<th>Contains</th>
<th>(Check) (one or more)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Tag</td>
</tr>
<tr>
<td>Attributes</td>
<td>none</td>
</tr>
</tbody>
</table>

**Check**

The `<Check>` element is used to specify connection capacity between specified members required for checking connections.

Table 68: Check model details

<table>
<thead>
<tr>
<th>Contains</th>
<th>n/a (empty element)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contained by</td>
<td>Checks</td>
</tr>
<tr>
<td>Attributes</td>
<td>Type (required, string), Desc (required, string), EquationID (required, string) - reference to equation IDs.</td>
</tr>
</tbody>
</table>
### Use of Wild Cards

Instead of specifying the capacities and other details for each individual beam or column section, "wild card" entries can be used. This is particularly useful when the remaining attributes are the same for multiple sections.

Example using the Default wild card:

```xml
<Beam Name="UB203x102x23">
  <BeamOrCol Name="Default" Mz.cap="35.9156" Fx.cap="207.3451" Fz.cap="" />
</Beam>
```

### Equations Guidelines

The following table contains the expressions that can be evaluated for user-defined equations:

<table>
<thead>
<tr>
<th>Digits/Characters</th>
<th>0</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>.</th>
<th>,</th>
</tr>
</thead>
<tbody>
<tr>
<td>Binary Operators</td>
<td>+</td>
<td>-</td>
<td>*</td>
<td>/</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unary Operators</td>
<td>+</td>
<td>-</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Parenthesis</td>
<td>(</td>
<td>)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Symbols</td>
<td>PI</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Functions</td>
<td>ABS</td>
<td>POW</td>
<td>ROUND</td>
<td>SQRT</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>MAX</td>
<td>MIN</td>
<td>SIN</td>
<td>COS</td>
<td>TAN</td>
<td>ASIN</td>
<td>ACOS</td>
<td>ATAN</td>
<td>LN</td>
<td>LOG</td>
<td>EXP</td>
<td></td>
</tr>
<tr>
<td>Reserved Keywords</td>
<td>[MZ.CAP]</td>
<td>[FX.CAP]</td>
<td>[FZ.CAP]</td>
<td>[FX]</td>
<td>[FY]</td>
<td>[FZ]</td>
<td>[MX]</td>
<td>[MY]</td>
<td>[MZ]</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Notes:
- "," (comma) can only be used with the POW function
- The ROUND function rounds off to the nearest integer, not to a certain decimal place
- Condition attributes should include any one of the following:
  - EQ - equal to
  - LT - less than
- LE - less than or equal to
- GT - greater than
- GE - greater than or equal to
- Reserved keywords should be specified with square brackets
- Reserved keywords, symbols, functions, and conditions are not case sensitive.

Example of an equation:

```
<Equation EquationID="Eq1" Equation="Abs([MZ])/[MZ.CAP]+Abs([FX])/[FX.CAP]"
    Condition="LT" Limit="1.0" />
```

Related Links
- D. Sample Connection Tags XML File (on page 994)

**D. Sample Connection Tags XML File**

This example file is installed in %LocalAppData%\Bentley\Engineering\STAAD.Pro CONNECT Edition\Default\Plugins\ConnectionTagLink. You can copy this file and use it as a template. Copies can be saved in any location and loaded through the user interface.

```xml
<?xml version="1.0" encoding="utf-8" ?>
<ConnectionTagFile>
  <FileVersion value="1.0" />
  <Categories>
    <Category CategoryName ="MOMENT">
      <CategoryDesc>End Moment Connection</CategoryDesc>
    </Category>
    <Category CategoryName ="SHEAR">
      <CategoryDesc>Single Shear Connection</CategoryDesc>
    </Category>
  </Categories>
  <Equations>
    <Equation EquationID="Eq1" Equation="(abs([Mz])+(alpha)*abs([Fx]))/[Mz.cap]"
      Condition="LT" Limit="1.0" />
    <Equation EquationID="Eq2" Equation="abs([Fy])/[Fy.cap]" Condition="LT"
      Limit="1.0" />
    <Equation EquationID="Eq3" Equation="(abs([Fy])+(alpha)*abs([Fx]))/[Fy.cap]"
      Condition="LT" Limit="1.0" />
  </Equations>
  <Tags>
    <Tag TagName ="EM" CategoryName ="MOMENT">
      <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
      <Capacities UnitSystem ="METRIC">
        <Beam Name="UB203x102x23">
          <BeamOrCol Name="UC152x152x23" Mz.cap="35.9156214893831" Fy.cap="207.345115136926" Fz.cap="" alpha="0.09695" />
          <BeamOrCol Name="UC152x152x30" Mz.cap="35.9156214893831" Fy.cap="207.345115136926" Fz.cap="" alpha="0.09695" />
          <BeamOrCol Name="UC152x152x37" Mz.cap="35.9156214893831" Fy.cap="207.345115136926" Fz.cap="" alpha="0.09695" />
          <BeamOrCol Name="UC203x203x46" Mz.cap="35.9156214893831" Fy.cap="207.345115136926" Fz.cap="" alpha="0.09695" />
          <BeamOrCol Name="UC203x203x60" Mz.cap="35.9156214893831" Fy.cap="207.345115136926" Fz.cap="" alpha="0.09695" />
          <BeamOrCol Name="UC254x254x73" Mz.cap="35.9156214893831" Fy.cap="207.345115136926" Fz.cap="" alpha="0.09695" />
        </Beam>
      </Capacities>
    </Tag>
  </Tags>
</ConnectionTagFile>
```
Design
D. Steel Design
Design
D. Steel Design

```xml
<Checks>
  <Check Type="MOMENT" Desc="Moment Check" EquationID="Eq1" />
  <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
</Checks>

<Tag TagName="EMH" CategoryName="MOMENT">
  <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
  <Capacities UnitSystem="METRIC">
    <Beam Name="UB203x102x23">
      <BeamOrCol Name="UC152x152x23" Mz.cap="80.1535632590318" Fy.cap="207.345115136926" Fx.cap="0" alpha="0.19695" />
      <BeamOrCol Name="UC152x152x30" Mz.cap="80.1535632590318" Fy.cap="207.345115136926" Fx.cap="0" alpha="0.19695" />
      <BeamOrCol Name="UC152x152x37" Mz.cap="61.8354487089892" Fy.cap="207.345115136926" Fx.cap="0" alpha="0.19695" />
      <BeamOrCol Name="UC203x203x46" Mz.cap="62.3614827257097" Fy.cap="207.345115136926" Fx.cap="0" alpha="0.19695" />
    </Beam>
  </Capacities>
</Tag>

<Tag TagName="SS" CategoryName="SHEAR">
  <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="1" MZ="1" />
  <Capacities UnitSystem="METRIC">
    <Beam Name="UB203x102x23">
      <BeamOrCol Name="Column" Mz.cap="" Fy.cap="77.0472" Fx.cap="" alpha="1" />
      <BeamOrCol Name="UB203x102x23" Mz.cap="" Fy.cap="53.3250236369366" Fx.cap="" alpha="0.692108520970738" />
      <BeamOrCol Name="UB203x102x30" Mz.cap="" Fy.cap="42.947706898342" Fx.cap="" alpha="0.557420735579515" />
    </Beam>
  </Capacities>
</Tag>
```

STAAD.Pro 996 User Manual
Design
D. Steel Design

STAAD.Pro 998 User Manual
**D. Connection Tags sub menu**

These sub-menu items are available under **Connection Tags** on the right-click pop-up menu when a member is selected.

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assign</td>
<td>Opens the <strong>Assign Connection Tags</strong> dialog and <strong>New Connection tag</strong> dialog, which are used to assign existing connection tags to member ends in the model and to create new connection tags, respectively.</td>
</tr>
<tr>
<td>Remove</td>
<td>Opens the <strong>Assign Connection Tags</strong> dialog and initiates the remove connection tag tool for the connections on the selected member. A confirmation dialog opens to confirm the remove action.</td>
</tr>
<tr>
<td>View</td>
<td>Opens the <strong>Assign Connection Tags</strong> dialog.</td>
</tr>
<tr>
<td>Check</td>
<td>(Active only after a successful analysis has been performed) Opens the <strong>Assign Connection Tags</strong> dialog and <strong>Check Connection Tags</strong> dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.</td>
</tr>
</tbody>
</table>

**Related Links**
- [D. Connection Tags XML File Schema](#) (on page 986)
- [G.9 Connection Tags](#) (on page 2333)
D. Concrete Design

D. Available Concrete Design Codes

The following concrete design codes are available in the batch design mode and in the interactive design mode.

Batch Design

Table 69: Concrete design codes available in Batch design

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Australia</td>
<td>AS 3600-2001 (on page 1627)</td>
<td>Australian Standard - Concrete Structures</td>
</tr>
<tr>
<td>Canada</td>
<td>CSA A23.3 1994 (on page 1696)</td>
<td>Design of Concrete Structures</td>
</tr>
<tr>
<td>China</td>
<td>GB50010-2002</td>
<td>Code for Design of Concrete Structures</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note</strong>: Design per the Chinese concrete code GB50010-2002 must be performed per the STAAD SSDD application, available in Chinese. Please download and install this application from Bentley SELECT or check with your Bentley account manager to see if this application is available in other languages.</td>
</tr>
<tr>
<td></td>
<td>IS 13920 2016 (on page 1955)</td>
<td>Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces - Code of Practice</td>
</tr>
<tr>
<td>Japan</td>
<td>AIJ 1991 (on page 1969)</td>
<td>Architectural Institute of Japan Standards for Structural Calculation of Steel Reinforced Concrete Structures</td>
</tr>
<tr>
<td>Mexico</td>
<td>NTC 1987 (on page 2002)</td>
<td>Normas Técnicas Complementarias para Diseño y construcción de Estructuras de Concreto</td>
</tr>
</tbody>
</table>
Concrete Design Workflow

The following design codes are available in the interactive concrete design workflow.

Table 70: Concrete Design codes available in STAAD.Pro RC Designer

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Australia</td>
<td>AS 3600-2001</td>
<td>Australian Standard - Concrete Structures</td>
</tr>
<tr>
<td>Canada</td>
<td>CAN/CSA A23.3-04 (R2010)</td>
<td>Canadian Standard A23.3-04 (Reaffirmed 2010)</td>
</tr>
<tr>
<td>China</td>
<td>GB50010</td>
<td></td>
</tr>
<tr>
<td>France</td>
<td>BAEL 91</td>
<td>Regles techniques de conception et de calcul des ouvrages et constructions en beton armé, suivant la méthode des états limites</td>
</tr>
</tbody>
</table>
### Design

**D. Concrete Design**

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>India</td>
<td>IS 456-2000 (on page 1055)</td>
<td>Code of Practice for Plain and Reinforced Concrete</td>
</tr>
<tr>
<td>Japan</td>
<td>AIJ 1985 (on page 1062)</td>
<td>Architectural Institute of Japan</td>
</tr>
<tr>
<td>Norway</td>
<td>NS 3473 (on page 1064)</td>
<td>Concrete structures - Design rules (Norwegian) (3rd Edition)</td>
</tr>
<tr>
<td>Russia</td>
<td>SP52-101-03 (on page 1066)</td>
<td>Concrete and Reinforced Concrete Structures Without Presstressed Reinforcement</td>
</tr>
<tr>
<td>Singapore</td>
<td>CP65 (on page 1071)</td>
<td>Code of Practice for Structural use of concrete</td>
</tr>
<tr>
<td>United Kingdom</td>
<td>BS 8110:1997, 1985 (on page 1081)</td>
<td>Structural use of concrete - Part 1: Code of practice for design and construction and Part 2: Code of practice for special circumstances (used for torsion design) and Amendments 1, 2, and 3</td>
</tr>
<tr>
<td>United States</td>
<td>ACI 318-05 (on page 1096)</td>
<td>Building Code for Structural Concrete</td>
</tr>
<tr>
<td></td>
<td>ACI 318-99 (on page 1086)</td>
<td>Building Code for Structural Concrete</td>
</tr>
</tbody>
</table>

**Related Links**

- *D. To specify concrete design code and parameters* (on page 1002)
- *D. Batch Design versus Interactive Design Workflows* (on page 944)

**D. Batch Member and Element Design Operations**

How to perform concrete member and slab design in the Analytical Modeling workflow using batch operations.

**D. To specify concrete design code and parameters**

To initiate the design of concrete members and specify the code parameters, use the following procedure.

**Note:** To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

1. On the **Analysis and Design** ribbon tab, select **Concrete** in the **Design** group gallery. The **Concrete Design - Whole Structure** dialog opens.
2. In the **Concrete Design - Whole Structure** dialog, select the applicable concrete design code from the **Current Code** drop-down list.

3. Click **Define Parameters**.
   The **Design Parameters** dialog opens.

4. Specify a value or option for each required parameter for a set of members and then click **Add**.
   You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.

   **Note:** Different parameters can be used for different member type designs (e.g., columns, beams, etc.). Alternatively, you can use a separate set of parameters for different member types.

5. Click **Close**.
   The design parameters are marked with an 📚 icon. This indicates that the need to be assigned to members.

6. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.

**Related Links**
- [D. Available Concrete Design Codes](on page 1000)
- [Concrete Design - Whole Structure dialog](on page 3085)
- [Design Parameters dialog](on page 3083)
- [TR.53.1 Design Initiation](on page 2859)
- [TR.53.2 Concrete Design-Parameter Specification](on page 2859)
- [TR.53.5 Concrete Design Terminator](on page 2861)

**D. To specify concrete beam design command**

To specify the design commands for concrete beams, use the following procedure.

1. On the **Concrete Design - Whole Structure** dialog, click **Commands**.
   The **Design Commands** dialog opens.
2. To perform a design action for concrete members, select one of the following options based on the member type:
   
   To... Select...
   
   design members acting as beams the DESIGN BEAM tab
   design members acting as columns the DESIGN COLUMN tab

3. Click Add.
4. Click Close.
5. Assign the design commands to the appropriate members.

Related Links
- Concrete Design - Whole Structure dialog (on page 3085)
- Design Commands dialog (on page 3084)
- TR.53.3 Concrete Design Command (on page 2860)

D. To generate concrete take off

To generate a summary of the total concrete volume along with the reinforcing steel bar numbers and weights for the design concrete members, use the following procedure.

1. On the Concrete Design - Whole Structure dialog, click Commands. The Design Commands dialog opens.
2. Select the **TAKE OFF** tab.
3. Click **Add**.
4. Click **Close**.
5. Assign the take off commands to members or to a named group of members.

**Related Links**
- *Concrete Design - Whole Structure dialog* (on page 3085)
- *Design Commands dialog* (on page 3084)
- *TR.53.4 Concrete Take Off Command* (on page 2860)

**D. Interactive Concrete Design**

A STAAD.Pro module.

**Note:** Interactive Concrete Design was referred to as "RC Designer" in older versions of STAAD.Pro.

**D. General Overview**

The STAAD.Pro Reinforced Concrete Designer (Concrete Design) is a post processor that works from the STAAD.Pro analysis results. The data is passed into Concrete Design includes the geometry, section and material properties, loads and combination information and analysis results.

For information on other components of the STAAD.Pro environment, refer to separate documentation.
D. Introduction

STAAD.Pro Concrete Design is started by selecting the menu option from STAAD.Pro. When this is done a link file is produced that contains the basic data for creating concrete designs. This data is created each time the program is started. Additional data that is created during the use of the program is stored, such as the members, envelopes, design groups and design briefs is stored in a persistent file. This means that if the program is closed and re-opened at a later date, the data remains available and does not need to be re-entered.

**Note:** At any time the persistent data can be removed by selecting the re-load link option from the File menu.

Related Links

- D. To open model in the interactive concrete design workflow (on page 1008)

D. Suitable Member Properties

Concrete beam and column designs can only be performed on physical members that have been created from beam elements (not plate or solid elements).

Concrete slab designs can only be performed that have been defined on slabs that have been created from finite element or plate elements.

The type of brief also requires that the beams are created from the following section property commands:

**Beam Design Briefs**

Rectangular or Square sections

<table>
<thead>
<tr>
<th>PRISMATIC</th>
<th>YD</th>
<th>ZD</th>
</tr>
</thead>
</table>

or

<table>
<thead>
<tr>
<th>User Table</th>
<th>PRISMATIC</th>
</tr>
</thead>
</table>

Tee shaped sections

<table>
<thead>
<tr>
<th>PRISMATIC</th>
<th>YD</th>
<th>ZD</th>
<th>YB</th>
<th>ZB</th>
</tr>
</thead>
</table>

or

<table>
<thead>
<tr>
<th>User Table</th>
<th>TEE</th>
</tr>
</thead>
</table>

**Column Design Briefs**

Rectangular or Square sections

<table>
<thead>
<tr>
<th>Prismatic</th>
<th>YD</th>
</tr>
</thead>
</table>

or

<table>
<thead>
<tr>
<th>User Table</th>
<th>PRISMATIC</th>
</tr>
</thead>
</table>

Circular sections

<table>
<thead>
<tr>
<th>Prismatic</th>
<th>YD</th>
</tr>
</thead>
</table>

(No User Table)
**Slab Design Briefs**

Quadrilateral or triangular finite elements defined in

```
ELEMENT  INCIDENCES (SHELL)
```

or

```
DEFINE  MESH
```

with suitable element thicknesses defined

```
ELEMENT  PROPERTY
(element list)  THICKNESS (value)
```

**D. File Types**

These are the files that are used by STAAD.Pro Concrete Design for a given STAAD.Pro project are:

<table>
<thead>
<tr>
<th>File Extension</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>.std</td>
<td>Structure Input File</td>
</tr>
<tr>
<td>.ben</td>
<td>Analysis Result File</td>
</tr>
<tr>
<td>.bmd</td>
<td>Analysis Result File</td>
</tr>
<tr>
<td>.dbs</td>
<td>Analysis Result File</td>
</tr>
<tr>
<td>.dsp</td>
<td>Analysis Result File</td>
</tr>
<tr>
<td>.rea</td>
<td>Analysis Result File</td>
</tr>
<tr>
<td>.scn</td>
<td>Analysis Result File</td>
</tr>
<tr>
<td>.sec</td>
<td>Analysis Result File</td>
</tr>
<tr>
<td>.Rei_Concrete</td>
<td>Persistent File</td>
</tr>
<tr>
<td>.Rei_Concrete_Link</td>
<td>Link file</td>
</tr>
</tbody>
</table>

All these files are stored in the same folder as the input file (.std).

The program also makes use of SProRC21.00.00.ini in the
C:\ProgramData\Bentley\Engineering\STAAD.Pro  CONNECT Edition folder. This file contains the user settings for fonts, colors, the current units, the conversion factors for the various unit settings.

**D. Using the Interactive Concrete Design**

This section contains some of the fundamental tasks and concepts used in Concrete Design.

**Tip:** It is also strongly recommended that you work through the example provided (on page 4936) to familiarize yourself with the program.
D. To open model in the interactive concrete design workflow

Concrete Design is a module of STAAD.Pro and as such needs data first created in the STAAD.Pro environment. Therefore to use Concrete Design, first create a suitable model in Analytical Modeling or Physical Modeling workflow in STAAD.Pro. This model includes the geometry, properties, loading, and analysis commands. The model must then be successfully analyzed.

1. In the STAAD.Pro User Interface, select Concrete Design in the Workflows panel.

The STAAD.Pro RC Designer window opens with the current model loaded.

**Note:** Concrete Design cannot be run as an independent application. It must be initiated from within STAAD.Pro.

**Related Links**
- D. Introduction (on page 1006)

D. Load Cases, Combinations, and Envelopes

STAAD.Pro and Concrete Design makes use of primary load cases, load combinations and load envelopes. The designs that are performed in RC Designer can make use of one or more sets of these results, but it is dependent upon the type of Design Brief that is to be used.

**Members Designed using Beam Design Briefs**

These perform designs on a defined envelope of primary load cases and/or load combinations. Currently, the envelope must be defined in the Envelope Page of the Design Layer Mode. The design is performed at a number of sections along each design member to satisfy the maximum positive and negative forces that exist at that section from all the load cases in that envelope.

**Members Designed using Column Design Briefs**

These perform designs on a collection of primary load cases and/or load combinations. Unlike the Beam Design Briefs, the design is done for the forces in each loadcase / combination and a suitable reinforcement layout defined accordingly.

**Floor Slabs Designed using Slab Design Briefs**

These perform designs on a defined envelope of primary load case and/or combinations. The design will report the reinforcement requirements of the maximum positive and negative moments at each location.

D. Design of Members and Slabs

Unlike analysis models which requires models to be broken down into separate analytical components of beam segments and finite elements, the designs in Concrete Design are performed on design members and slabs.

The design members and slabs need to be created in the Concrete Design environment.

Design members can be designed with either Beam or Column Design Briefs.
D. Rules for Forming Members

To form a member, the following rules must be obeyed.

- All elements must form a single continuous line. But they do not have to form a straight line. Thus curved members may be formed.
- There must be a free end. Whilst curved members are allowed, they cannot form a closed loop.
- All elements must have the same beta angle.
- All elements must point in the same direction. Check with the orientation labels if necessary. Use the reverse element command on elements that point the wrong way.
- None of the elements can be part of another member.
- The section properties must be consistent at each element end. Elements can taper along their length, but where one element ends and the next starts, they must have the same section reference.
- All elements must be made from the same material.

D. Creating a design (physical) member

1. Select the Des. Layer | Members mode (on page 1202).
2. Select the Elements Cursor tool (default selection tool).
3. Select the constituent element(s) that will combine to form a design (physical) member

   **Tip:** Hold the <Ctrl> key to select a number of elements.

   **Note:** Refer to D. Rules for Forming Members (on page 1009) for element rules when forming a physical member.

   **Note:** An analytical beam segment can only be part of one design member definition.

4. Select either:
   - Members > Form Member
   - The Form Member tool.

   Information on Design Members can be selected using the Members Cursor. Details of design members can be seen in the Members Table (on page 1151).

D. Deleting a design member

1. Select the Des. Layer | Members mode.
2. Select the Members Cursor tool.
3. Either:
   - Press <Delete>.
   - or
Select **Edit > Delete.**

**D. Rules for Forming Slabs**

To form a slab, the following rules must be obeyed.

- All elements must form a single continuous area.
- None of the elements can be part of another Slab.
- All elements must be made from the same material.

**D. Creating a design slab**

1. Select the **Des. Layer | Slabs page** (on page 1202).
2. Select the **Select Plates** cursor tool.

3. Select the constituent element(s) that combine to form a design slab.

   **Tip:** Hold the <CTRL> key to select a number of elements.

   **Note:** Refer to **D. Rules for Forming Slabs** (on page 1010) for element rules when forming a slab.

   **Note:** A finite element can only be part of one slab definition.

4. Either:
   - Select **Slabs > Form Slab.**
   - The **Form Slabs** tool.

Information on design slabs that have been created is listed in the **Slabs Table** (on page 1153).

**D. Deleting a design slab**

1. Select the **Des. Layer | Slabs mode.**
2. Select the **Select Slabs** tool.
3. Either:
   - Press <Delete>.
   - Select **Edit > Delete.**

**D. Design Briefs**

Designs are performed on design members and design slabs according to a set of rules defined in the appropriate code. A number of the criteria, such as the dimensions of the beams, columns, and slabs have
already been defined in the STAAD.Pro model. However, certain aspects have not, such as the grade of concrete and steel reinforcing bars. All the parameters that are required are collected together and referred to as a Design Brief. Each Design Brief is tailored to the requirements of the governing design code.

Details of contents of each brief can be found in section 4.3.5 (on page 1218).

**Available Codes**

c_design_brief_dialogs (on page 1218) for information on the dialogs for a specific member type and code.

<table>
<thead>
<tr>
<th>Country/Region</th>
<th>Code</th>
<th>Design Type</th>
<th>Page</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>United States</td>
<td>ACI 318-99</td>
<td>Beam (on page 1218)</td>
<td>Column (on page 1221)</td>
<td>Column (on page 1221)</td>
</tr>
<tr>
<td></td>
<td>ACI 318-05 / 318M-05</td>
<td>Beam (on page 1224)</td>
<td>Column (on page 1230)</td>
<td>Slab (on page 1235)</td>
</tr>
<tr>
<td>Japan</td>
<td>AIJ</td>
<td>Beam (on page 1239)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>France</td>
<td>BAEL</td>
<td>Beam (on page 1247)</td>
<td>Column (on page 1252)</td>
<td></td>
</tr>
<tr>
<td>Great Britain</td>
<td>BS 8110</td>
<td>Beam (on page 1255)</td>
<td>Column (on page 1257)</td>
<td>Slab (on page 1260)</td>
</tr>
<tr>
<td>Singapore</td>
<td>CP65</td>
<td>Beam (on page 1269)</td>
<td>Column (on page 1271)</td>
<td></td>
</tr>
<tr>
<td>Germany</td>
<td>DIN 1045-1</td>
<td>Beam (on page 1274)</td>
<td>Column (on page 1280)</td>
<td></td>
</tr>
<tr>
<td>Europe</td>
<td>Eurocode 2 - 2004</td>
<td>Beam (on page 1286)</td>
<td>Column (on page 1290)</td>
<td></td>
</tr>
<tr>
<td>Egypt</td>
<td>ECCS 203</td>
<td>Beam (on page 1301)</td>
<td>Column (on page 1304)</td>
<td></td>
</tr>
<tr>
<td>Spain</td>
<td>EHE</td>
<td>Beam (on page 1307)</td>
<td>Column (on page 1313)</td>
<td></td>
</tr>
<tr>
<td>India</td>
<td>IS456</td>
<td>Beam (on page 1325)</td>
<td>Column (on page 1328)</td>
<td></td>
</tr>
<tr>
<td>Norway</td>
<td>NS3473</td>
<td>Beam (on page 1331)</td>
<td>Column (on page 1337)</td>
<td></td>
</tr>
<tr>
<td>Turkey</td>
<td>TS 500</td>
<td>Beam (on page 1347)</td>
<td>Column (on page 1350)</td>
<td></td>
</tr>
<tr>
<td>Russia</td>
<td>SP52-101-03</td>
<td>Beam (on page 1342)</td>
<td>Column (on page 1344)</td>
<td></td>
</tr>
</tbody>
</table>
### D. Design Groups

The final stage in preparation is collecting together the design members and slabs into a Design Group. The Design Group contains not only the list of members (or slab number) to be designed, but also which Design Brief that is to be used.

The list of Design Groups can be seen in the Design Groups table (on page 1160). This has separate tabs for Member Groups and Slab Groups.

#### D. Creating a Member Design Group

1. Either:
   - Select the Des. Layer | Groups/Briefs page.
   - or
   - Select Groups > Design Group Table.

   The Design Group table (on page 1160) opens.

2. Select the Members tab on the Design Groups table.
3. Select the members to be included in the design group using the Member Cursor tool.

   **Note:** A design member may be part of more than one design group.


   The New Design Group dialog (on page 1149) opens.

5. (Optional) Specify a name for the Design Group.
6. Select a previously created Design Brief to be used for designing the physical elements in this group.
7. Click OK.
D. Modifying the design members included in a Design Group

1. Select the group number in the Design Groups table.
2. Click on Edit Design Groups at the bottom of the table.
   The Group Member List table (on page 1160) opens.

D. Adding a member to the group

1. Select the last (empty) cell in the table.
2. Enter the member number.
3. Press either the <Tab> or <Enter> key.

D. Adding members to an existing design group

1. Select one or more members in the View window.
2. Right click to display the View window pop-up menu.
3. Select Add member(s) to Design Group...
   The Add Member(s) to Design Group dialog box opens.
4. Select the Design Group to which you want to add the selected members.

   Tip: The current Design Group is selected by default.
5. Click the Add button.
6. Click OK to close the dialog.

D. Removing a member from a group

1. Select the member number in the table.
2. Press <Delete>.

D. Coping a reinforcing cage design for re-use in another column group

Note: If the member has a reinforcement cage already defined, then the program checks the reinforcement for compliance. Otherwise the program will select a reinforcement cage from the bars and specifications of the Design Brief to minimize the area of main reinforcement bars.

1. Select a column member included in the previously designed column group.
2. Select Edit > Copy.
   Reinforcement cage details are copied to the clipboard.
3. Select columns to which you want to assign the reinforcing details.

   Note: Target columns must be of the same dimensions as the source column.
4. Select Edit > Paste.
   The details are copied to the target columns and can then be checked for compliance.

D. Creating a Slab Design Group
1. Select the Slabs tab on the top of the Design Groups Table
2. Click the New Design Slab button.
   The New Design Slab dialog (on page 1162) opens.

3. (Optional) Specify a name the Design Group
4. Select a previously created Slab Design Brief to be use for designing the slab.
5. Select the physical slab number for this Design Slab.
   
   **Note:** Slab design groups only contain a single slab, but this can be divided into separate Regions later.

6. Click OK.

D. Using View Windows

View windows are used to display all or parts of the structural model. Most model object selection is made using the mouse pointer (through various object cursor tools) in the view windows.

Multiple view windows may be open simultaneously. The top view window is referred to as the active view window. Actions which affect view windows are typically applied only to the active view window.

Views are manipulated by the view and zoom tools found on the Standard toolbar (on page 1193) or by the Orientation dialog (on page 1133).

**Tip:** View Windows have context-sensitive pop-up menus. When different items are selected (or even when there is no selection), right-clicking will display commonly used functions for the current selection set.
**D. Creating a new view of a portion of the model**

**Tip:** This feature is useful for manipulating portions of larger structures.

1. Select one or more objects in the active view window.
2. Select View > New View
   - The **New View** dialog opens prompting if you want to replace the current active window or open a new active window.
3. Select a window option and click OK.
   - The new view is opened displaying on the selected model object(s).
D. Saving a view window for later use

You can save a view window which displays <untitled> in the window title bar.

1. Select View > View Management > Save View
   The Save View As dialog (on page 1136) opens.
2. Type a title for the view.
3. Click OK.

D. Reopening a saved view

1. Select View > Open View.
   The Open View dialog (on page 1134) opens.
2. Select a previously saved view from list.
3. Select if the view is re-open in a view window or to replace the active view window.
4. Click OK.
   The view window re-opens and is now the active view window.

Related Links
- D. Open View dialog (on page 1134)

D. Report Setup

The graphical user interface (GUI) will report the summary of a design and display the summary of calculations performed. Each design brief also has its own detailed calculation routines that allow the important details of the calculations to be viewed.

To view the detailed calculations, select the Design Detail for inclusion in the Report Setup dialog (on page 1110).

Note: If the design group has a large number of members, this may produce a very large document. Therefore, you may want to select only a few members for reporting with this option.

Pictures can also be added to a report, including symbolic elevations and sections of the reinforcement.

D. Creating a symbolic picture

1. Click anywhere on the diagram or cross section segments of the reinforcement in a Main Reinforcement Diagram
2. Select the Take Picture tool.
   The Picture Setup dialog box (on page 1016) opens. of the diagram, which can be resized to fit within the print space.

This is added to the available items in the Repost Setup and can be selected and added to the Selected items as with any other print item.

D. Picture Setup dialog
Used to create a reinforcing detail picture for inclusion in reports. The file contains elevations, sections and a bending schedule.
Design
D. Concrete Design

Opens when the **Take Picture** tool is selected for a Main Reinforcing Diagram or a Shear Reinforcement Diagram.

The controls in this dialog are analogous to those in the Export Drawing dialog (on page 1188).

**OK**  
Accepts the changes made and closes the dialog.

**Cancel**  
Closes the dialog without making any changes.

D. Design or Check

Concrete Design can function in one of two ways, either as a Design or as a Check.

The Design Method is the default in which the design process will run through and provide the minimum reinforcement required based on the associated design brief.

The Check Method is available if you change the reinforcement and selects **Design > Design Now**. With this method, the specified reinforcement is checked at all the locations and the status for the overall design member is reported.

**Caution:** This method is only available for designs performed on design groups that have a column design brief.
D. Earthquake Checks

For select design codes, Concrete Design can perform additional earthquake detailing checks per the building code.

D. To Perform a Preliminary Design per EC2 in RC Designer

In order for EC8 checks to be performed, a member must have passed all EC2 checks in the initial design step. To also help identify issues in design or detailing, if one step of checks fails for a member, further checks will not be performed on that member.

The program will first check that all materials in the design brief are satisfactory. Then, moment capacity of beams and columns are evaluated and compared. EC8 stipulates that columns must have a moment capacity of 1.30 times the sum of the moment capacity of beams framing at that joint. This ensures beams are the initial failure mode.

**Note:** Deflection checks are performed for individual members only.

   - The following National Annexes are available in RC Deigner:
     - Singapore National Annex
     - UK National Annex
     - Malaysian National Annex
   - You can also select **EC2 without Annex** and specify the NA parameters manually.

2. Specify Medium (DCM) or High (DCH) in the Ductility level list.
   - Low ductility (DCL) does not require additional Eurocode 8 checks and therefore is not included.

3. Specify the load case containing seismic loads for use with design checks.

4. Specify the Structural Type in the list.
   - Currently, only Framed/ Dual/Coupled Wall Systems are supported in RC Designer.

5. Select if you wish to Check reinforcement for detailing rules for ductility per EC8.

6. Specify a Curvature ductility factor to be used for column checks.
   - If not specified, the default value is taken as unity (1).

7. Click Check Now to perform all design and detailing checks.

**Related Links**
- [P. Using the Earthquake Workflow](on page 2240)

D. To perform seismic design and detailing per EC8

To perform seismic design and detailing checks per Eurocode 8, use the following procedure.

A preliminary design per Eurocode 2 must be made for all concrete members prior to detailing and design per Eurocode 8. In order for EC8 checks to be performed, a member must have passed all EC2 checks in the initial design step.

1. Select the **Earthquake** tab.
   - The **Collapse Check Setup** dialog opens.
2. Select **EC8 - 2004** from the **Design Code** list. Currently, only the UK Annex is available.

3. Select the **Ductility level** of the structure:
   - **DCM** - Medium ductility
   - **DCH** - High ductility
   
   **Note**: Low ductility (DCL) does not require additional Eurocode 8 checks and therefore is not included.

4. Select the **Earthquake load case** containing seismic loads for use with design checks.

5. Specify the **Structural Type** in the list. Refer to the code for descriptions of each type.

6. Check the option to **Check reinforcement detailing rules for ductility** if applicable.

7. Specify a **Curvature ductility factor** to be used for column checks. If not specified, the default value is taken as unity (1).

8. Click **Check Now**. The program performs the design and detailing checks.

To help identify issues in design or detailing, if one step of checks fails for a member, further checks will not be performed on that member.

The program will first check that all materials in the design brief are satisfactory. Then, moment capacity of beams and columns are evaluated and compared. EC8 stipulates that columns must have a moment capacity of 1.30 times the sum of the moment capacity of beams framing at that joint. This ensures beams are the initial failure mode.

**Note**: Deflection checks are performed for individual members only.

**Related Links**
- *P. Using the Earthquake Workflow* (on page 2240)
- *P. Using the Earthquake Workflow* (on page 2240)

**D. Engineering Theory**

This section contains technical reference data on the methodology behind concrete design used in the Concrete Design workflow.
D. Design Messages

The design is performed on the selected members from the current Design Group and the parameters in the associated Design Brief. Each section is designed and a reinforcement cage is identified. The results of all the sections are then considered and a final rationalized cage is defined that requires the minimum area of reinforcement, but still matching all the design requirements including spacing and cracking checks.

If there are any issues arising form the design that results in either a partial or fully incomplete design for one or more of the design members, then these are reported in a Design Messages dialog at the end of the design of all of the members. Note that on models where a large number of members are being designed, with a large number of load cases, then the member that is currently being designed is displayed on the lower left side of the status bar.

On completion of the design, the members are color-coded to indicate their status. The colors can be set using the Colour Manager dialog (on page 1132).

D. Design Codes

The information that the design uses is determined by the design code associated with the design group.

D. Australian Concrete design per AS 3600:2001

Two design briefs for the Australian concrete design code AS 3600: 2001 have been included in the Concrete Design. The design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>Description</th>
</tr>
</thead>
</table>

D. AS 3600 Beam Design Principles

Design is performed per AS 3600-2001. Steel Strength reduction factors are taken from Table 2.3. Characteristic strength of materials are used based on section 6

Strength of Beams

Beams are designed for flexure, and shear. The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Lateral bending can be considered if user selects the option. Compression reinforcement is provided where required.

Flexure design is done based on the rectangular stress block defined in clause 8.1.2.2

\[ \gamma = [0.85 = 0.007(f_c - 28)] \]

Minimum Strength requirement as per 8.1.4

\[ \frac{A_{sd}}{b \cdot d} \geq 0.22 \left( \frac{D}{d} \right)^{2} \frac{f'c}{f_{sf}} \]

Requirements for shear reinforcement Clause 8.2.5

Where \( V^* \leq \phi V_{uc} \), no shear reinforcement is required, except that where the overall depth of the beam exceeds 750 mm, minimum shear reinforcement shall be provided in accordance with Clause 8.2.8

Where \( V^* > \phi V_{u,min} \). Shear reinforcement shall be provided in accordance with clause 8.2.10

Shear strength limited by web crushing: Maximum shear force resisted by element as per the clause 8.2.6
Design

D. Concrete Design

\[ V_{u,\text{max}} = 0.2 \, f'_c b_v d_0 \]

Shear Strength of a beam excluding shear reinforcement

\[ V_{uc} = b_1 b_2 b_3 d_0 (A_{st} f'_c / b_v d_0)^{1/3} \]

Strength of beams in shear 8.2 section

Related Links

- **D. AS 3600 Beam Design Brief dialog** (on page 1241)
- **D. AS 3600 Column Design Brief dialog** (on page 1244)

D. AS 3600 Column Design Principles

Design has been carried out based on the code provisions in AS 3600-2001. Steel Strength reduction factors are taken from Table 2.3. Characteristic strength of materials are used based on section 6.

1. Design procedure carried out based on clause 10.2, 10.2.1 and 10.2.3.
2. Short column design based on 10.3
3. Design of slender columns based on 10.4
4. Minimum bending moment not less than \( N^* \) times 0.05\( D \), where \( D \) is the overall depth of the column in the plane of the bending moment.
5. Radius of gyration may be taken as 0.3\( D \), where \( D \) is the overall dimension in the direction in which stability is being considered and for circular cross-section, \( r \) may be taken as 0.25\( D \).
6. Biaxial bending and compression based on clause 10.6.5
7. Reinforcement requirements for columns: 10.7.1 Limitations on longitudinal steel:
   a. less than 0.01\( A_g \)
   b. Not exceed 0.04\( A_g \)

Related Links

- **D. AS 3600 Beam Design Brief dialog** (on page 1241)
- **D. AS 3600 Column Design Brief dialog** (on page 1244)

**D. Canadian Concrete Design per CAN/CSA-A23.3-04 (R2010)**

Two design briefs for CAN/CSA-A23.30-04 (R2010) (including updates 1, 2, and 3) are available in the Concrete Design. The design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>Design of Concrete Structures</td>
</tr>
<tr>
<td>Revision</td>
<td>2010 (Reaffirmed year by CSA)</td>
</tr>
</tbody>
</table>

D. CAN/CSA-A23.3-10 Beam Design Principles

The CAN/CSA-A23.3-10 Beam Design Brief is for single or multi-span, prismatic, rectangular solid, or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to **D. Suitable Member Properties** (on page 1006) for more details.

Beams are designed for flexure, torsion, and shear only. Each member is divided into equally spaced sections along its length and the design is performed at each of these locations. You can specify the number of segments to be considered (between 4 and 25) for each span. Sections are also taken at the face of supports and at locations of the maximum positive and negative bending moments.

Design for Flexure
The main (longitudinal) reinforcement is calculated for both sagging and hogging moments (i.e., positive and negative moment, respectively) on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of the design forces, and thus, at each of the defined sections the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment values from the analysis.

The beam is then divided into sub-beams, those that can use the same reinforcement cage and having

- a. Same overall beam size
- b. Same cover requirements

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement pattern is calculated for the range of bars specified in the Design Brief. The program does not have a limit on the number of bars in any one layer as long as the spacing requirements specified in the code are satisfied. The program can handle a maximum of four layers of reinforcement, two each for the top and bottom layers.

The program then performs a check at each of the defined sections to determine the number of bars, if any, that can be cut off. The reinforcement bars will not be curtailed at these sections in the following cases:

- a. If the bars are required for compression, or
- b. If curtailing these bars would result in a failure of crack-width checks, or
- c. If curtailing these bars would result in a failure of the minimum reinforcement checks.

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, $F_{z}$, acting on the beam. The minor axis shear forces are not considered in the design.

The bar size for the shear links and the minimum number of shear legs to be provided are specified in the Design Brief. Therefore, the required spacing for minimum links can be defined. The program then checks each section to determine the shear force $V_{Ed}$ and concrete shear capacity $V_{Rd,c}$. From this, the section is classified as either a minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $\nu$, and concrete shear capacity, $\nu_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.
If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

![Diagram](image)

*Figure 121: Minimum shear links required, V, for shear forces between these values*

**Code Clauses Implemented**

The requirements of CAN/CSA-A23.3-10 utilized in the design module are as follows:

**Materials**

8.4.2-3
8.5.1
8.6.1.1-2
8.6.4-5
10.1.3
10.1.5-6

**Minimum Reinforcement**

7.4.1.1
10.5.1.1
10.5.1.3
10.6.2

**Service Limit States**

10.6.1

**Ultimate Limit States**

10.1.2
10.5.2
11.2.8.1-2
11.2.9.1
11.2.10
11.3.1
11.3.3-4
11.3.5.1
11.3.6.4
11.3.8.1
Design

D. Concrete Design

11.3.8.3
11.3.9.2-5
11.3.10.2-6

Ductility

10.5.2

Development Length

12.1.2
12.2.1-2
12.2.4-5
12.3.1-3
12.5.1-3

Related Links

- D. CSA-A23.3-10 Beam Design Brief dialog (on page 1262)

D. CAN/CSA-A23.3-10 Column Design Principles

The CAN/CSA-A23.3-10 Column Design Brief is for prismatic, solid rectangular, or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered. Therefore, only single height columns are recommended.

Design Principles

Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Provided reinforcement is calculated by the program based on the forces generated from the analysis model and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then rechecked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the “Guiding Load Case”, and the bar arrangement is chosen based on that combination. The detailed printout of the design calculations is for that combination.

Code Clauses Implemented

The requirements of CAN/CSA-A23.3-10 utilized in the design module are as follows:

Materials

8.4.2-3
8.5.1 – Design calculations are based on the specified yield strength of reinforcement. A warning is reported if the specified yield strength is greater than 500 MPa.
8.6.1.1
8.6.2.2 – The modulus of elasticity of concrete is calculated using equation 8-1 in the code. The specified density, $\gamma_c$, is used in the equation but if it is outside of the range 1,500 to 2,500 kg/m$^3$, a warning is reported. Altern
8.6.4-5 – The modulus of rupture, $f_r$, is calculated use equation 8-3.
10.1.3
10.1.5-6

Minimum Reinforcement
7.4.1.1
7.4.1.3
7.6.5.1-2 – Bundled bars are not considered.
10.9.1-4 – The minimum reinforcement for columns is 0.01×A_g. If reinforcement exceeds 0.04×A_g, a warning is reported. If reinforcement exceeds 0.08×A_g, a failure is reported. A_g is the gross cross sectional area of the column. A minimum of 4 or 6 bars is required for rectangular or round columns, respectively (for irregularly shaped columns, no design check is performed).
10.10.5

Ultimate Limit State

10.1.2 - The strain in the reinforcement and concrete are assumed to be directly proportional to the distance from the neutral axis. For deep flexural members, the design is performed using the plane sections remain plane assumption, but a warning is reported.
10.1.5 – The tensile strength of concrete is neglected.
10.9.4
10.10.4-5
11.2.8.1-2
11.2.9.1
11.2.10
11.3.1
11.3.3-4
11.3.5.1
11.3.6.4
11.3.8.1
11.3.8.3
11.3.9.2-3
11.3.10.1-6

Development Length

12.1.2
12.2.1-2
12.2.4-5
12.3.1-3
12.5.1-3

Related Links

• D. CSA-A23.3-10 Column Design Brief dialog (on page 1265)

D. Chinese Concrete design per GB50010

Code Ref GB50010-2002
Description Chinese Standard, Concrete Structures, GB50010
Revision 2002

Release 2007
D. Selected Specifications
Chinese concrete design specifications to achieve the following specifications:

- *Concrete Structure Design Code* (GB 50010-2002)
- *Technical specification for concrete high-rise building* (JGJ 3-2002)
- *Structural Load code for the design of building structures* (GB 50009--2001 (2006))
- *Seismic Design Code of Buildings* (GB 50011-2001)

Unless specifically stated in this document, its provisions are derived from standard reference design of concrete structures (GB 50010-2002).

## Nomenclature

### Material Properties

- $E_c$: elastic modulus of concrete;
- $E_{fc}$: Concrete Fatigue Deformation modulus;
- $E_s$: modulus of elasticity of reinforced;
- $C_{20}$: that the standard cube strength of concrete strength value 20N/mm$^2$;
- $f'cu$: a side length of 150mm concrete cube compressive strength of the construction phase;
- $f_{cu, k}$: a side length of 150mm cube of concrete compressive strength of standard value;
- $f_{ck}$, $f_c$: standard value of concrete compressive strength, design values;
- $f_{tk}$, $f_t$: the standard value of axial tensile strength of concrete, the design value;
- $f'_{ck}$, $f'tk$: Construction of the concrete axial compression, axial tensile strength of standard value;
- $f_{yk}$, $f_{ytk}$: in reinforced, prestressed reinforced strength standard value;
- $f_y$, $f'_{y}$: ordinary steel tensile and compressive strength of the design value;
- $f_{py}$, $f'_{py}$: prestressed reinforcement tensile, compressive strength design value.

### Capacity and Resistance

- $N$: design value of axial force;
- $N_k$, $N_q$: according to the standard combination of load effects, quasi-permanent combination of computing the value of the axial force;
- $N_{ux}$, $N_{uy}$: axial force on the X axis, Y axis of the eccentric or eccentric compression tension design value of bearing capacity;
- $M$: bending moment design value;
- $M_k$, $M_q$: according to the standard combination of load effects, quasi-permanent combination of calculated bending moment;
- $M_u$: Component design flexural capacity value;
- $M_{cr}$: Bending cross section of the cracking moment value;
- $T$: torque design value;
- $V$: shear design values;
- $V_{cs}$: oblique section of concrete and stirrups on shear strength design values;
- $F_l$: local load design value or the concentration of counter-force design value.
σck, σcq the standard combination of load effects, quasi-permanent combination of checking the
dge of the concrete crack under normal stress;
σpc generated by the pre-tension concrete normal stress;
σtp, σcp concrete principal tensile stress, principal stress;
σfc, max, σfc, min fatigue tension zone when Checking the edge of fiber reinforced concrete compression
zone or the maximum stress, minimum stress;
σs, σp is set in the vertical plane bearing capacity of ordinary steel, prestressing stress;
σsk according to the standard combination of load effects calculated longitudinal tensile steel
stress or equivalent stress;
σcon tensioning control stress;
σp0 prestressed concrete method to force the point of zero stress when the stress of
prestressed reinforcement;
σpe effective prestress prestressed reinforcement;
σl, σ'l tension zone, compression zone in the corresponding stage of prestressed reinforced the
value of prestress loss;
τ shear stress of concrete;
ωmax according to the standard combination of load effects and to consider the impact of long-
term effect of the maximum crack width calculation.

Geometric Parameters

a, a' longitudinal tensile reinforcement force point, longitudinal compressive force point to sections of
reinforced near-edge distance;
as, a's longitudinal non-prestressed tensile reinforcement force point, the longitudinal non-prestressed
compression steel sections together to point to the near edge of the distance;
ap, a'p area of longitudinal prestressing tension force point, the compression zone of longitudinal
prestressing force points to the section near the edge distance;
b width of rectangular section, T-shaped, I-shaped cross-section of the web width;
bf, b'f T-shaped or I-shaped cross section in tension zone, compression zone of the flange width;
d the diameter of the bar diameter or cross section;
c the thickness of concrete;
e, e' axial force point to point longitudinal tensile reinforcement force, vertical compression
reinforcement force point distance;
e0 axial force on the section center of gravity of eccentricity;
ea Add eccentricity;ei initial eccentricity;
h Height of the Cross;
h0 effective height of cross section;
hf, h'f T-shaped or I-shaped cross section in tension zone, compression zone of the flange height;
Design

D. Concrete Design

i section radius of gyration;
rc radius of curvature;
la longitudinal tensile reinforcement of the anchorage length;
l0 Beam Calculation of span or length of columns;
s component axis along the direction of the spacing of transverse reinforcement, the spacing of spiral reinforcement or stirrups spacing;
x concrete compression zone;
y0, yn conversion section focus, focus to the calculation of the net section of the fiber distance;
z longitudinal tensile reinforcement force to the concrete compression zone the distance between the points together;
A component-sectional area;
A0 component conversion section area;
An member net section area;
As, A's tension zone, compression zone of non-prestressed longitudinal cross-sectional area;
Ap, A'p tension zone, compression zone cross-sectional area of longitudinal prestressing;
Asv1, Ast1 in shear, by the calculation of single-leg stirrups twisted cross-section area;
Astl Torsion calculation of longitudinal torsion access to all the cross-sectional area of non-prestressed reinforcement;
Asv, Ash limb within the same section of vertical, horizontal steel hoop or distribution of all cross-sectional area;
Asb, Apb the same plane bent non-tensioned, prestressed bent steel bar cross-sectional area;
Al Concrete local compression area;
Acor steel mesh, spiral reinforcement or stirrups within the inner surface of the concrete core area;
B Bending stiffness of the section;
W cross section by pulling the edge of the elastic resistance moment;
W0 transformed section by pulling the edge of the elastic resistance moment;
Wn net section tensile edge of the elastic resistance moment;
Wt section by twisting the plastic resistance moment;
I moment of inertia;
I0 transformed section moment of inertia;
In net section moment of inertia.

Calculation Coefficient and Other

α1 compression zone of concrete rectangular stress diagram of the stress value and design value of concrete compressive strength ratio;
αE reinforced concrete elastic modulus and elastic modulus ratio;
Concrete strength coefficient;  
$\beta_1$ rectangular stress block compression zone and the neutral axis height (axis in the compression zone to the edge of the distance) ratio;  
$\beta_l$ Local Compressive concrete strength coefficient;  
$\gamma$ the section of concrete to resist the plastic moment of impact factor;  
$\eta$ eccentric compression effects of the second order moment of the axial force eccentricity magnification factor;  
$\lambda$ Calculation of cross sections of shear span ratio;  
$\mu$ friction coefficient;  
$\rho$ the reinforced longitudinal reinforcement ratio;  
$\rho_{sv}, \rho_{sh}$ vertical stirrups, stirrups, or the vertical distribution of horizontal bar, horizontal distribution of steel reinforcement ratio;  
$\rho_v$ indirectly, the volume of steel reinforcement ratio or the hoop;  
$\varphi$ stability factor of axial compression members;  
$\theta$ consider the long-term effect on the deflection load increases the impact factor;  
$\psi$ crack between the longitudinal bar strain coefficient of uniformity.

Concrete Materials

1. Elastic modulus of concrete calculated as follows (note the provisions of 4.1.5):  
$$E_c = \frac{10 \times 10^5}{2.2 + \frac{24.7}{f_{cu,k}}}$$

2. Standard values of concrete compressive strength and design values were calculated the following formula (note the provisions of 4.1.3):  
$$f_{ck} = 0.88\alpha_c f_{cu,k}$$  
$$f_c = f_{ck}/Y_c = f_{ck}/1.4$$  
C50 and the following take $\alpha_c1 = 0.76$, on the C80 take $\alpha_c1 = 0.82$, according to the linear law of change in the middle.  
Of the C40 take $\alpha_c2 = 1.0$, on the C80 take $\alpha_c1 = 0.87$, change in the middle by a linear law.  

3. The standard value and design of axial tensile strength values were calculated as follows (note the provisions of 4.1.4):  
$$f_{tk} = 0.88\left(0.395\right)f_{cu,k}^{0.55}\left(1 - 1.645\delta\right)^{0.45}\alpha_c2$$  
$$f_c = f_{tk}/Y_c = f_{tk}/1.4$$

Reinforcement Methods

1. Concrete grades available
Table 72: Table 4.1.4 Design value of concrete strength 表 4.1.4 混凝土强度

<table>
<thead>
<tr>
<th>Strength Category</th>
<th>Concrete Strength (N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>C15</td>
</tr>
<tr>
<td>fc</td>
<td>7.2</td>
</tr>
<tr>
<td>fi</td>
<td>0.91</td>
</tr>
</tbody>
</table>

2. Steel grades available for reinforcement

Table 73: Table 4.2.3-1 Ordinary strength of reinforcing bar design value 表 4.2.3-1 普通箍筋强度

<table>
<thead>
<tr>
<th>Classification</th>
<th>Symbol</th>
<th>fy</th>
<th>fy'</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hot-rolled steel</td>
<td>Φ</td>
<td>210</td>
<td>210</td>
</tr>
<tr>
<td>HRB 335 (20MnSi)</td>
<td>Φ</td>
<td>300</td>
<td>300</td>
</tr>
<tr>
<td>HRB 400 (20MnSiV, 20MnSiNb, 20MnTi)</td>
<td>Φ</td>
<td>360</td>
<td>360</td>
</tr>
<tr>
<td>400 (K20MnSi)</td>
<td>R</td>
<td>360</td>
<td>360</td>
</tr>
</tbody>
</table>

Related Links
- D. GB50010 Beam Design Brief dialog (on page 1318)
- D. GB50010 Column Design Brief dialog (on page 1321)

Limit State Design

For the ultimate limit state, structural members shall be the basic combination of load effects or accidental combination of the following limit state design equation:

\[ \gamma 0 S \leq R (3.2.3-1) \]

where

- \(\gamma 0\) = importance factor, input by the user in the program, on the security level for the one or the design life of more than 100 years and structural components, should not be less than 1.1; on the security level for the design life span of two or 50 years of structural members, not less than 1.0; on the security level is 3 or the design life of 5 years and below the structural members, not less than 0.9; in seismic design, structural elements do not take into account the importance of factors;

- \(S\) = the ultimate state of the load effect combination of the design value.

- \(R\) = design value of bearing capacity of structural members; in seismic design, seismic bearing capacity divided by the adjustment factor \(\gamma RE\);

For the limit state, structural components should be separately according to the standard combination of load effects, quasi-permanent combination or standard combination, and to consider long-term effects of limit state design with the following expression:
Design

D. Concrete Design

\[ S \leq C \ (3.3.1) \]

where

\[
S = \text{limit state of load effect combination of value;}
\]

\[
C = \text{structural members meet requirements under normal use of the deformation, crack width and stress limits.}
\]

D. Design Capacity of Beams

Beams along the bearing capacity of beams through a long span of each cross section, beam cross section of the design include a component of the capacity of checking and regulating the provision of construction or other design requirements such as thickness reinforcement, reinforcement ratio, bar diameter.

Related Links

- D. GB50010 Beam Design Brief dialog (on page 1318)
- D. GB50010 Column Design Brief dialog (on page 1321)

General Provisions

1. Flexural capacity is calculated according to the following fundamental assumptions:
   a. section to maintain plane strain;
   b. does not take into account the tensile strength of concrete;

2. Compressive stress-strain relationship of concrete in accordance with the provisions of Article 7.1.2-3 standard values.
   a. Ultimate compressive strain of concrete is in accordance with section following formula:

   \[
   \varepsilon_{cu} = 0.0033 - (f_{cu,k} - 50) \times 10^{-5}
   \]  
   (7.1.2-5)

   where

   \[
   \varepsilon_{cu} = \text{cross section of the concrete ultimate compressive strain, while in the non-uniform compression, press formula (7.1.2-5) calculations, such as the calculation of } \varepsilon_{cu} \text{ is greater than 0.0033, is taken as 0.0033; when in axial When pressure is taken as } \varepsilon_0; \]

   \[
   f_{cu,k} = \text{concrete cube compressive strength of standard value}
   \]

   \[
   \beta_1 = \text{coefficient, when the concrete strength when no more than C50, } \beta_1 \text{ taken as 0.8, when the concrete strength grade of C80 time, } \beta_1 \text{ taken as 0.74, during determined by linear interpolation.}
   \]

   \[
   \alpha_1 = \text{rectangular stress diagram of the stress is taken as design value of concrete compressive strength } f_c \text{ multiplied by the coefficient } \alpha_1. \text{ When the concrete strength when no more than C50, } \alpha_1 \text{ taken as 1.0, when the concrete strength grade of C80 time, } \alpha_1 \text{ taken as 0.94, during determined by linear interpolation.}
   \]

3. Longitudinal tension and compression zone of reinforced concrete damage yield simultaneously, the relative height of compression zone limits \( \xi_b \) By the following formula:

   \[
   \xi_b = \frac{\beta_1}{1 + \frac{f_y}{E_s f_{cu}}}
   \]  
   (7.1.4-1)

Net Concrete Cover Thickness

Input by the user of longitudinal force ordinary reinforced and prestressed steel, the concrete cover thickness (reinforced concrete surface to the outer edge of the distance) should not be less than the nominal diameter of steel, and shall comply with Table 9.2.1 of GB50010-2002 requirements.

General Provisions on Seismic Design
Seismic fortification intensity level and
Provide beam seismic rating, fortification intensity, structure type and component type of input. Program based on user input seismic fortification intensity levels and components of the beam design and construction inspection.

Seismic bearing capacity adjustment factor
Consider the combination of earthquake concrete structure, its cross-section bearing capacity of the internal forces generated by seismic load bearing capacity divided by the seismic design value adjustment factor $\gamma_{RE}$. Seismic bearing capacity adjustment factor $\gamma_{RE}$ 11.1.6 in accordance with GB50010-2002 form used. For the beam component, $\gamma_{RE}$ value is 0.75.

Calculation of Concrete Beams
Bearing Capacity
1. Rectangular Bending Capacity: rectangular cross section or flange in the tension side of the inverted T-shaped cross-section flexural members, the flexural capacity should meet the following requirements:

   \[ M \leq \alpha_1 f_c b x (h_0 - x/2) + f_y A'_s (h_0 - \alpha'_s) \]  

   Concrete Compression Zone according to the following formula:

   \[ \alpha_1 f_c b x = f_y A'_s + f_{py} A_p \]  

   Concrete Compression Zone still should meet the following conditions:

   \[ x \leq \xi_b h_0 \]  

   \[ x \geq 2\alpha'_s \]  

2. When included in the calculation of longitudinal compression reinforcement general, should meet the GB50010-2002 formula (7.2.1-4) conditions; if not satisfied this condition, the flexural capacity should meet the following requirements:

   \[ M \leq f_{py} A_p (h - a_p - a'_s) + f_y A_s (h - a_s - a'_s) + (\sigma'_p - f_{py}) A'_p (a'_p - a'_s) \]  

3. Bending Flexural bearing capacity calculations, as required by the structure or by checking Limit State requirements configured sectional area of longitudinal tensile steel bending capacity requirements greater than the area of reinforcement, the calculation of concrete compression Height $x$, only the conditions included in the required bending capacity vertical cross-section area of tensile steel.

4. T-Section Bending Capacity: flange in the compression zone of the T shape, I shaped cross-section flexural members (Figure 7.2.2), its flexural capacity should meet the following requirements:

   a. When the following conditions are met

      \[ f_y A_s + f_{py} A_p \leq \alpha_1 f_c b' (h_0 - h' f/2) + f_y A'_s (h_0 - \alpha'_s) + (\sigma'_p - f_{py}) A'_p \]  

      Should be a width $b'$ rectangular cross section;

   b. When satisfied equation (7.2.2-1) the conditions

      \[ M \leq \alpha_1 f_c b x (h_0 - x/2) + \alpha_1 f_c (b' f - b) h' f (h_0 - h' f/2) + f_y A'_s (h_0 - \alpha'_s) + (\sigma'_p - f_{py}) A'_p (h_0 - \alpha'_p) \]  

   Concrete Compression Zone according to the following formula:

   \[ \alpha_1 f_c b x + (b' f - b) h' f = f_y A_s + f_y A'_s + f_{py} A_p + (\sigma'_p - f_{py}) A'_p \]  

Concrete compression zone height limit of seismic
For the seismic component: in the calculation, included in the longitudinal compression reinforcement in concrete compression zone of beam end shall meet the following requirements:

A seismic level
ξb ≤ 0.25 \quad \text{(11.3.1-1)}

Second, three seismic rating
ξb ≤ 0.35 \quad \text{(11.3.1-2)}

Reinforcement ratio test

1. Configuration bar when the protective layer thickness in accordance with user input calculated as; to complete the inspection section bearing capacity of reinforced configuration, when the actual use of force reinforced the role of location as a point of steel.

2. Flexural members, offset tension, axial tension component side of the minimum tensile steel reinforcement ratio of 0.2 and $45f_t/f_y$ the larger value;

3. The seismic structure:

<table>
<thead>
<tr>
<th>Seismic Level</th>
<th>Beam Position</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Bearing</td>
</tr>
<tr>
<td>1</td>
<td>0.4 and $80f_t/f_y$ in the large value</td>
</tr>
<tr>
<td>2</td>
<td>0.3 and $65f_t/f_y$ in the large value</td>
</tr>
<tr>
<td>3,4</td>
<td>0.25 and $55f_t/f_y$ in the large value</td>
</tr>
</tbody>
</table>

Beam, the lower the minimum longitudinal steel reinforcement ratio, non-seismic design should not be less than 0.30%, respectively; seismic design, special one, one and two respectively, not less than 0.60%, 0.50% and 0.40%; (JGJ 10.2.8-1)

4. The largest component seismic reinforcement rate: end of the beam longitudinal reinforcement ratio of tension reinforcement should not exceed 2.5%.

Reinforcing the principles and methods of allocation

1. As selected through a long beam under the critical section, thus setting the upper and lower steel reinforcement combinations.

A. Through long tendons meet the upper

i. At least two, the corner bar.

ii. Of the secondary seismic level, the bar diameter of not less than 14mm, and should not be less than the beam, respectively top and bottom ends of the longitudinal section of the reinforced area of the larger 1 / 4; on 3, 4 seismic rating, bar diameter not less than 12mm.

iii. The first row of steel from the non-pass a long column (beam) side has been extended to ln / 3 position; second row extend ln / 4 position; ln value is defined as: for end bearing, net-based cross-span; For the intermediate bearing, ln on both sides for the bearing of a large net across the span.

iv. Reinforced concrete longitudinal beam support section of negative moment reinforcement in tension should not be cut off in the tension zone. When to cut, in accordance with the provisions of reinforcement anchorage length (GB50010-2002 9.3.1), retained after anchoring length cut off under stress conditions.

2. Through the lower part of a long bar to meet:

i. At least two, the corner bar.
ii. When the earthquake, the full reach into bearing.

iii. Non-seismic, do not reach into the lower bearing of the beam longitudinal reinforcement cut off dot pitch bearing edge distance 0.1ln.

Test Oblique Section

Program based on user input information to carry out shear reinforcement component of shear resistance test, specified as follows:

Non-Seismic Shear Resistance Test

1. Rectangular, T-shaped and I-shaped cross-section flexural members, the shear section shall meet the following conditions:

   When the $h_w / b ≤ 4$:
   \[ V ≤ 0.25β_c f_c bh_0 \]

   When the $h_w / b ≥ 6$:
   \[ V ≤ 0.2β_c f_c bh_0 \]

   Combination of box beam cross section the maximum shear design values shall meet the following requirements:
   \[ V ≤ 0.2β_c f_c bh_0 \]

2. Not configured stirrups and bent sheet metal reinforcement general bending members, the diagonal section of the shear capacity shall meet the following requirements:

   \[ V ≤ 0.7β_h f_t bh_0 \]

   \[ β_h = (800 / h_0) ^ {1 / 4} \]

3. Rectangular, T-shaped and I-shaped cross-section flexural members in general, when the only configuration stirrups, its diagonal section of the shear capacity shall meet the following requirements:

   \[ V ≤ 0.7f_t bh_0 + 1.25 f_{yy} (A_{sv} / s) h_0 \]

Seismic shear resistance test

1. Consider the combination of the framework of the seismic design of beam end shear value $V_b$ according to the actual value;

2. Consider the combination of the frame beam earthquake, when the inter-high ratio of $l0 / h > 2.5$, its shear cross-section should conform to the following conditions:

   \[ V_b ≤ 1 (0.20β_c f_c bh_0) / γRE \]  \hspace{1cm} \text{(11.3.3)}

   where

   \( β_c = \) concrete strength factor: When the concrete strength less than C50, the take $β_c = 1.0$; when the concrete strength grade of C80, the take $β_c = 0.8$; during determined by linear interpolation.

3. Consider the combination of earthquake rectangular, T-shaped and I-shaped cross-section of the frame beam, the Shear capacity should meet the following requirements:

   \[ V_b ≤ 1 [0.42f_t bh_0 + 1.25 f_{yy} A_{sv} h_0 / s] / γRE \]

Tie Ratio Test

1. Confining rate: When $V > 0.7f_t bh_0$, the hoop reinforcement ratio $\rho_{sv} [\rho_{sv} = A_{sv} / (b_s) (b_s)]$ not be less than $0.24f_t / f_{yy}$;

2. Seismic conditions:
Along the beam length of stirrup reinforcement ratio $\rho_{sv}$ should meet the following requirements:

**Seismic level 1:**

$$\rho_{sv} \geq 0.30 \frac{f_t}{f_{yv}}$$

**Seismic level 2:**

$$\rho_{sv} \geq 0.28 \frac{f_t}{f_{yv}}$$

**Seismic level 3,4:**

$$\rho_{sv} \geq 0.26 \frac{f_t}{f_{yv}}$$

3. High-rise building structures, seismic rating for the particular level of reinforced concrete structures shall meet the basic requirements of a seismic level, the beam end encryption District hoop structure confining rate increases of 10%.

4. Box beam supports of encrypted area with hoop hoop smallest, non-seismic design should not be less than 0.9$f_t/f_{yv}$; seismic design, the special one and one and two respectively, not less than 1.3$f_t/f_{yv}$, 1.2$f_t/f_{yv}$ and 1.1$f_t/f_{yv}$. ([JGJ 10.2.8 - 2])

**Spacing**

1. The maximum stirrup spacing:

   a. Non-seismic:

   **Table 75: The maximum beam stirrup spacing (mm), Table 10.2.10**

<table>
<thead>
<tr>
<th>Beam h</th>
<th>$V&gt;0.7f_t bh_0$</th>
<th>$V\leq0.7f_t bh_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>150 &lt; h ≤ 300</td>
<td>150</td>
<td>200</td>
</tr>
<tr>
<td>300 &lt; h ≤ 500</td>
<td>200</td>
<td>300</td>
</tr>
<tr>
<td>500 &lt; h ≤ 800</td>
<td>250</td>
<td>350</td>
</tr>
<tr>
<td>h &gt; 800</td>
<td>300</td>
<td>400</td>
</tr>
</tbody>
</table>

When the beam with longitudinal compression by computing needs reinforcement, the stirrups should be made of closed, then ($d$ is the minimum diameter of longitudinal compression reinforcement):

<table>
<thead>
<tr>
<th>Condition</th>
<th>Maximum Distance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Beam with longitudinal compression computing needs according to the time bar</td>
<td>$\min(15d,400)$</td>
</tr>
<tr>
<td>Pressure within the longitudinal layer of steel more than 5 and greater than 18mm in diameter</td>
<td>10d</td>
</tr>
</tbody>
</table>
### Condition

<table>
<thead>
<tr>
<th>Condition</th>
<th>Maximum Distance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Beam width is greater than 400mm and a layer of pressure within the longitudinal reinforcement than three, or when the beam width of not more than 400mm but a layer of reinforcement within the vertical pressure when more than 4</td>
<td>Composite hoop bar should be set</td>
</tr>
</tbody>
</table>

The spacing of non-encrypted area of the spacing should not be greater than the encryption area 2 times.

b. Seismic encryption District: meet the requirements of Table 6.3.2-2, and a seismic rating, not more than 20 times 200mm and stirrups diameter is greater; 2, 3 seismic level, not more than 20 times 250mm and hoop tendon diameter is greater; 4 seismic rating, not more than 300mm.

2. Encryption area: seismic design, the beam end encryption area stirrup length should conform to the requirements of Table 6.3.2-2; beam end set the framework of the first node from the edge of stirrups should be no larger than 50mm.

**Table 76: Frame beam stirrups beam end zone construction requirements encryption, Table 11.3.6-2**

<table>
<thead>
<tr>
<th>Seismic Level</th>
<th>Encryption Zone Length (mm)</th>
<th>maximum stirrup spacing (mm)</th>
<th>Stirrup minimum diameter (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2h and 500 of the larger value</td>
<td>6 times the longitudinal bar diameter, high beam, 1 / 4 and 100 in the minimum</td>
<td>10</td>
</tr>
<tr>
<td>2</td>
<td>1.5h and 500 of the larger value</td>
<td>8 times the longitudinal bar diameter, high beam, 1 / 4 and 100 in the minimum</td>
<td>8</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>8 times the longitudinal bar diameter, high beam, 1 / 4 and 150 in the minimum</td>
<td>8</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>8 times the longitudinal bar diameter, high beam, 1 / 4 and 150 in the minimum</td>
<td>6</td>
</tr>
</tbody>
</table>

**Note:** H as the height of the table section.

3. Box beam supports of (left side column height within the beam section 1.5) stirrups should be encrypted, encrypted area stirrup diameter of not less than 10mm, spacing should not exceed 100mm. *(JGJ-10.2.8-3)*

Minimum diameter stirrups:

1. # Non-seismic design: A cross-section height h > 800mm of the beam, the stirrup diameter not less than 8mm; on the beam height h ≤ 800mm beam, the stirrup diameter not less than 6mm. Beam with vertical
D. Concrete Design

compression reinforcement computing needs, the stirrup diameter of not less than the vertical compression reinforcement should be 0.25 times the diameter.

2. Seismic design: minimum diameter shall meet the requirements of Table 6.3.2-2, when the end of the beam longitudinal reinforcement ratio greater than 2%, the minimum diameter of the table in the stirrups should be increased 2mm.

Side bar (waist bars) layout

When the beam web height \( h_w \geq 450\text{mm} \), the two sides of the beam height profile along the vertical structure should be reinforced, each side of the vertical structural steel (not including the beam, the lower part of the reinforced and erect steel) cross-section area of not web-sectional area should be less than \( 0.1\% bh_w \), and the pitch should be less than 200mm. Here, the web height \( h_w \) according to GB50010-2002 7.5.1 Article access.

D. Design Capacity of Columns

Concrete column design through the user-specified component type, seismic parameters such as standardized test grades, the program selected by the user load combinations and design parameters for the design and reinforcement, testing results and design process will be output to the report.

Related Links

- D. GB50010 Beam Design Brief dialog (on page 1318)
- D. GB50010 Column Design Brief dialog (on page 1321)

General Provisions

See “General Provisions” (on page 1031) for Beam Design.

Column Capacity Under Axial Load

Axial compression component capacity

Axially loaded reinforced concrete structures, when the configuration of the stirrup meet GB50010-2002 the provisions of section 10.3, which is cross-section compression capacity should meet the following requirements:

\[
N \leq 0.9\phi(f_cA + f_yA_{s'})
\]  
(7.2.15)

When the vertical steel reinforcement ratio greater than 3%, the formula (7.2.15) in \( A \) Should use \( (A-A') \) in its place.

Axially loaded reinforced concrete structures, when the configuration of the screw or welded steel ring indirectly found GB50010-2002 the provisions of section 10.3, which is cross-section compression capacity should meet the following requirements:

\[
N \leq 0.9(f_cA_{cor} + f_yA_{s'} + 2\alpha f_yA_{sat})
\]  
(7.3.2-1)

Eccentric compression component capacity

Eccentric rectangular cross section compression member compression capacity should meet the following requirements:

\[
N \leq \alpha_1 f_c bx + f_y A_{s'} - \sigma_y A_s
\]  
(7.3.4-1)

\[
N e \leq \alpha_1 f_c bx(h_0 - x/2) + f_y A_{s'}(h_0 - a_s)
\]  
(7.3.4-2)

Uniform configuration along the peripheral ring of longitudinal reinforcement section eccentric compression, which is cross-section compression capacity should meet the following requirements:

\[
N \leq a_1 f_c A(1 - \frac{\sin 2n\alpha}{2n\alpha}) + (a - a_1)f_yA_s
\]  
(7.3.7 - 1)

\[
N\eta e_i \leq a_1 f_c A \frac{\sin 3\alpha}{n} + f_y A_{s'} \frac{\sin 3\alpha + \sin n\alpha_1}{n}
\]  
(7.3.7 - 2)
Axial tension component capacity

Axial tension member tensile capacity of normal section shall meet the following requirements:

\[
N \leq f_y A_s
\]  \hspace{1cm} (7.4.1)

Eccentric tension component capacity

Eccentric rectangular symmetric reinforcement for tension members, both large and small offset tension of the situation, may according to formula (7.4.2-2) calculated:

\[
N_{e'} \leq f_y A_s (h'0 - a_s) + f_{py} A_p (h'0 - ap)
\]  \hspace{1cm} (7.4.2-2)

Calculation of Oblique Section

Eccentric compression

Rectangular reinforced concrete eccentric compression, the Shear Capacity meet the following requirements:

\[
V \leq \frac{1.75}{\lambda + 1} f_y b h_o + f_{yu} \frac{A_{sv}}{s} h_o + 0.07N
\]  \hspace{1cm} (7.5.12)

Shear span ratio calculation section the following provisions shall be drawn:

1. Columns on the kinds of structures, should take \(\lambda = M / (V h_o)\); on the frame structure of the frame column, when the inflection point in the story context, the desirability of \(\lambda = H_n / (2 h_o)\); when \(\lambda < 1\), take \(\lambda = 1\); when \(\lambda > 3\), the take \(\lambda = 3\); here, \(M\) for the calculation of the cross section and the shear design value \(V\) corresponding moment design value, \(H_n\) dear height for the column.

2. On the other eccentric compression, when the bear uniformly distributed load, take \(\lambda = 1.5\); when the bear meet GB50010-2002 section 7.3.4 provides a concentrated loads, taking \(\lambda = a / h_0\), when \(\lambda < 1.5\), the taking \(\lambda = 1.5\); when \(\lambda > 3\), the take \(\lambda = 3\); here, \(a\) for the concentrated load to the bearing or distance from the edge node.

Eccentric Tension Member

Eccentric rectangular cross-section of reinforced concrete tension members, the Shear Capacity meet the following requirements:

\[
V \leq \frac{1.75}{\lambda + 1} f_y b h_o + f_{yu} \frac{A_{sv}}{s} h_o + 0.07N
\]  \hspace{1cm} (7.5.14)

Seismic design of concrete columns

Longitudinal reinforcement force

Columns and pillars of steel frame configuration, should meet the following requirements

Box columns and pillars all vertical steel reinforcement percentage of the force should not be less than the values specified table 11.4.12-1 the same time, each side of the reinforcement ratio of not less than 0.2;

Box columns and pillars of strength in all the longitudinal reinforcement ratio should not exceed 5%. Column longitudinal steel should symmetric configuration. Column section size greater than 400mm, vertical spacing of reinforcement should not be greater than 200mm. When the level of design by an earthquake, and the column shear span ratio \(\lambda \leq 2\), the columns on each side of longitudinal steel reinforcement ratio should be less than 1.2%.
### Table 77: All the reinforced vertical column minimum reinforcement percentage (%), Table 11.4.12-1

<table>
<thead>
<tr>
<th>Column Type</th>
<th>Seismic Level</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
</tr>
<tr>
<td>Framework of columns, edge columns</td>
<td>1.0</td>
</tr>
<tr>
<td>Framework of the corner column, pillar box</td>
<td>1.2</td>
</tr>
</tbody>
</table>

**Note:** All of the reinforced column longitudinal reinforcement ratio of the minimum, when using HRB400 grade steel, the values in the table should be reduced by 0.1; when the concrete strength grade of C60 and above, shall be increased by 0.1 values in the table.

### Stirrups

Box columns and pillars of the upper and lower ends of stirrups should be encrypted, encrypted area of the stirrups and the stirrups minimum diameter maximum spacing should be consistent with the provisions of the table 11.4.12-2;

### Table 78: Column side stirrup encryption zone construction requirements, Table 11.4.12-2

<table>
<thead>
<tr>
<th>Seismic Level</th>
<th>Maximum Stirrup Spacing (mm)</th>
<th>Stirrup minimum diameter (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6 times the longitudinal bar diameter and 100 of the smaller value</td>
<td>10</td>
</tr>
<tr>
<td>2</td>
<td>8 times the longitudinal bar diameter and 100 of the smaller value</td>
<td>8</td>
</tr>
<tr>
<td>3</td>
<td>8 times the diameter of longitudinal reinforcement and 150 (column root of 100) of the small value</td>
<td>8</td>
</tr>
<tr>
<td>4</td>
<td>8 times the diameter of longitudinal reinforcement and 150 (column root of 100) of the small value</td>
<td>6 (column root 8)</td>
</tr>
</tbody>
</table>

**Note:** Column refers to the underlying root of column top of the basement or no basement, a basis for the top; column length should be taken root encrypted area of the layer of column clear height of not less than 1 / 3; when there are rigid surface, in addition to column-side stirrup Encryption should be rigid yet outside the ground, the height of 500mm under the stirrups within the encryption.
Pillar box and the shear span ratio $\lambda \leq 2$ the framework of the column should be encrypted column within the whole stirrup high, and the spacing should not exceed 100mm;

Two level columns earthquake, when the stirrup diameter is not less than 10mm, less than 200mm from the limb, in addition to the root column, the stirrup spacing should be allowed to use 150mm; three seismic level frame column section size not greater than 400mm, the stirrups should be allowed to use the minimum diameter 6mm; 4 seismic rating columns span ratio less than 2, the stirrup diameter is not less than 8mm.

Columns of the stirrup length encryption area, take the long side of column section size (or cross section diameter), column clear height of $1 / 6$ and 500mm in maximum. Secondary level prism earthquake along the whole column stirrups high encryption.

Encrypted column stirrup limbs from the region: a seismic level not greater than 200mm; two, three seismic levels should be less than 250mm in diameter and 20 times larger value of hoop; 4 seismic rating not greater than 300mm. In addition, every one vertical bar in both directions should have stirrups or stretching constraints.

One, two, three levels of various types of seismic structure of the frame column and pillar boxes, the axial load ratio $N / (f_c A)$ is not greater than the limits specified Table 11.4.16.

Table 79: Framework of the axial compression ratio, Table 11.4.16

<table>
<thead>
<tr>
<th>Structural System</th>
<th>Seismic Level</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
</tr>
<tr>
<td>Frame</td>
<td>0.7</td>
</tr>
<tr>
<td>Frame - shear wall, tubular structures</td>
<td>0.75</td>
</tr>
<tr>
<td>Part of the frame-supported shear wall</td>
<td>0.6</td>
</tr>
</tbody>
</table>

Notes:

a. An axial load ratio $N / (f_c A)$ that consider the combination of earthquake and the box frame column pillar design value of axial pressure on the whole sectional area of N and column A and the design value of concrete compressive strength $f_c$ ratio of the product; of the non-seismic the role of the structure calculation, to take no seismic design values combined axial force;

b. When the concrete strength grade of C65-C70, the axial compression ratio should decrease according to values in the table 0.05; concrete strength grade of C75-C80, the axial compression ratio values in the table should decrease by 0.10;

c. shear span ratio $\lambda \leq 2$ columns, the axial compression ratio should be values in the table decreases 0.05;

Column encryption area stirrup reinforcement ratio of the volume of stirrups meet the following requirements:

1. column volume of stirrups stirrup reinforcement encrypted area ratio, shall meet the following requirements:

$$\rho_v \geq \lambda \frac{f_c}{f_y} \quad (11.4.17)$$

where
\( \rho_v \) = column encryption area the size of stirrup reinforcement ratio, according to GB500 10-2002 section 7.8.3 of the Ordinance calculated, should be less overlap in the stirrups volume

\( f_c \) = design value of concrete compressive strength; when the intensity level of less than C35, the values according to C35;

\( f_{yv} \) = stirrups and reinforcement design value of tensile strength;

\( \lambda_v \) = Minimum stirrup values used in Table 11.4.17.

<table>
<thead>
<tr>
<th>Seismic Level</th>
<th>Stirrup Type</th>
<th>Axial Compression Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>≤0.3</td>
</tr>
<tr>
<td>1</td>
<td>General hoop, composite hoop</td>
<td>0.10</td>
</tr>
<tr>
<td></td>
<td>Spiral hoop, composite or continuous composite rectangular spiral hoop</td>
<td>0.08</td>
</tr>
<tr>
<td>2</td>
<td>General hoop, composite hoop</td>
<td>0.08</td>
</tr>
<tr>
<td></td>
<td>Spiral hoop, composite or continuous composite rectangular spiral hoop</td>
<td>0.06</td>
</tr>
</tbody>
</table>
### Seismic Level, Stirrup Type, and Axial Compression Ratio

<table>
<thead>
<tr>
<th>Seismic Level</th>
<th>Stirrup Type</th>
<th>Axial Compression Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>≤0.3</td>
<td>0.06 0.07 0.09 0.11 0.13 0.15 0.17 0.20 0.22</td>
</tr>
<tr>
<td>3</td>
<td>General hoop, composite hoop</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Spiral hoop, composite or continuous composite rectangular spiral hoop</td>
<td>0.05 0.06 0.07 0.09 0.11 0.13 0.15 0.18 0.20</td>
</tr>
</tbody>
</table>

#### Notes:

a. A general refers to a single rectangular hoop or single circular hoop; spiral hoop that a single spiral reinforcement; composite hoop means rectangular, polygonal, circular or reinforcement consisting of stirrups stirrup; compound spiral hoop means spiral hoop and rectangular, polygonal, circular or reinforcement consisting of stirrups stirrup; continuous composite rectangular spiral refers to all the spiral hoop to hoop with a steel processed into the stirrups;

b. In the calculation of the volume of composite spiral hoop reinforcement ratio, one of the spiral reinforcement of the volume of non-conversion factor should be multiplied by 0.8;

c. On 1, 2, 3, 4 seismic rating column, the encrypted area of stirrups stirrup reinforcement ratio of the volume should not be less than 0.8%, respectively, 0.6%, 0.4% and 0.4%;

d. Concrete strength higher than C60, the stirrup should the composite hoop, composite or continuous composite rectangular spiral hoop; when the axial compression ratio is not greater than 0.6, the encryption region the value of the minimum stirrup according to values in the table should increase 0.02; when the axial compression ratio greater than 0.6, the appropriate values in the table increased by 0.03.

2. Box pillar should adopt the compound spiral hoop or crosswise composite hoop, the minimum transverse reinforcement characteristic values should be the value in Table 11.4.17 0.02 increase access to, and the volume reinforcement ratio of not less than 1.5%;

3. When the shear span ratio λ ≤ 2, the one, two, three seismic level of composite column should spiral hoop or crosswise composite hoop, the hoop reinforcement ratio of volume of not less than 1.2%; 9, when the degree of fortification should not be less than 1.5%.

---

**D. Egypt Concrete design per ECCS 203**

Concrete Design has two ECCS 203 design briefs, the design principles utilized by these are defined in:
D. ECCS 203 Beam Design Principles

The ECCS 203 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are pre scanned to identify the critical load cases at different sections of the beams. The total number of sections considered is defined in the General sheet of the parameters dialog box. It can be set to between 4 and 25 segments (per span), thus will be checked at the ends of these segments, plus the locations of the maximum forces.

Slenderness Effects and Analysis Considerations

STAAD provides the user with two methods of accounting for the slenderness effects in the analysis and design of concrete members. The first method is equivalent to the procedure presented in ECCS203-2004 equation 4-11. In this section, the code recognizes that additional moments induced by deflection are present and states that these 'secondary' moments are accounted for by the design formula in equation 6-38,6-37 etc. This is the method used in the design for concrete in STAAD.

Alternatively STAAD contains a PDELTA ANALYSIS facility, which allows the effects of these second order moments to be considered in the analysis rather than the design. In a PDELTA analysis, after solving the joint displacements of the structure, the additional moments induced in the structure are calculated. These can be compared to those calculated using the formulation of ECCS203-2004.

Design for Flexure

Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the above mentioned sections. Each of these sections are designed to resist both of these critical sagging and hogging moments. Currently, design of singly reinforced sections only is permitted. If the section dimensions are inadequate as a singly reinforced section, such a message will be permitted in the output. Flexural design of beams is performed in two passes. In the first pass, effective depths of the sections are determined with the assumption of single layer of assumed reinforcement and reinforcement requirements are calculated. After the preliminary design, reinforcing bars are chosen from the internal database in single or multiple layers. The entire flexure design is performed again in a second pass taking into account the changed effective depths of sections calculated on the basis of reinforcement provided after the preliminary design. Final provision of flexural reinforcements are made then. Efforts have been made to meet the guideline for the curtailment of reinforcements as per ECCS203-2004. Although exact curtailment lengths are not mentioned explicitly in the design output (finally which will be more or less guided by the detailer taking into account of other practical consideration), user has the choice of printing reinforcements provided by STAAD at 13 equally spaced sections from which the final detailed drawing can be prepared.

Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. The shear checks are performed at the same locations along each span as for the bending check including the additional locations for the maximum shear forces amongst the active load cases and the associated torsional moments. Shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by STAAD. Two-legged stirrups are provided to take care of the balance shear forces acting on these sections.

Related Links
D. ECCS 203 Column Design Principles

The ECCS 203 Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

Design Principles

Columns are designed for axial force and biaxial bending at the ends. All active loadings are tested to calculate reinforcement. The loading which produces maximum reinforcement is called the critical load and is displayed. The requirements of ECCS203-2004 equation 6-37,6-38,6-41 etc are followed, with the user having control on the effective length parameters. Bracing conditions are controlled by using the BRACE parameter. The program will then decide whether or not the column is short or slender and whether it requires additional moment calculations.

Related Links

- D. ECCS 203 Beam Design Brief dialog (on page 1301)
- D. ECCS 203 Column Design Brief dialog (on page 1304)

D. European Concrete Design per Eurocode EC2: 2004

Two additional design briefs for Eurocode 2: 2004 have been included in the Concrete Design. The design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>Description</th>
<th>Revision</th>
</tr>
</thead>
</table>

D. Eurocode 2:2004 Beam Design Principles

The Eurocode 2:2004 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.

Beams are designed for flexure and shear only. Each member is divided into equally spaced sections along its length and the design is performed at each of these locations. The user can specify the number of segments to be considered (between 4 and 25) for each span. Sections are also taken at face of supports and at locations of the maximum positive and negative bending moments.

Design for Flexure

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of the design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment values from the analysis.

The beam is then divided into sub-beams, those that can use the same reinforcement cage and having

- Same overall beam size
- Same covers requirements
For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement pattern is calculated for the range of bars specified in the Design Brief. The program does not have a limit on the number of bars in any one layer as long as the spacing requirements specified in the code are satisfied. The program can handle a maximum of 4 layers of reinforcement, two each for the top and bottom layers.

The program then performs a check at each of the defined sections to determine the number of bars, if any, that can be curtailed. The reinforcement bars will not be curtailed at these sections in the following cases:

a. If the bars are required for compression or
b. If curtailing these bars would result in a failure of crack-width checks or
c. If curtailing these bars would result in a failure of the minimum reinforcement checks.

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, $F_z$ acting on the beam. The minor axis shear forces are not considered in the design.

The bar size for the shear links and the minimum number of shear legs to be provided are specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear force $V_{Ed}$ and concrete shear capacity $V_{Rd,c}$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $\sigma$, and concrete shear capacity, $\sigma_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.
Anchorage and Bond Lengths

Anchorage and bond lengths are calculated in accordance with the requirements of Eurocode 2. They can be displayed graphically on the main reinforcement diagram and are used for the schedule table and export to rebar detailing programs.

Code Clauses Implemented

The requirements of Eurocode 2 utilized in the design module are as follows:

2 Basis of Design

2.4.2.4 Partial factors for materials are taken from the National Annex

3 Material Properties

3.1.2.3 Implemented by the equations in table 3.1 where possible
3.1.6 Followed
3.1.7(3) Followed
3.2.3 fyk used
3.2.7(2) b Followed
3.2.7(4) Followed
3.2.4.3.(1) Followed

4 Durability and Cover to Reinforcement

4.4.1.2.(3) Followed

5 Structural Analysis

5.9. (3) Followed

6 Ultimate Limit States

6.1(1)-(3) Followed
6.2.1(1)-(5) Followed
6.2.1(7) Followed
6.2.2(1) Followed
6.2.3(1)-(3) Followed – Note 2 is ignored
6.2.3(7) Followed

8 Detailing of Reinforcement and Pre-stressing Tendons - general
9 Detailing of members and Particular rules

9.2.1.1(1) Followed
9.2.1.1(3) Followed
9.2.1.2(1) Followed
9.2.1.3(1) Followed
9.2.1.3(2) Followed
9.2.1.4(1) Followed
9.2.1.4(2) Followed
9.2.1.5(1) Followed
9.2.1.5(2) Followed
9.2.2(1) Followed (Always 90 degrees)
9.2.2(5) Followed
9.2.2(6) Followed
9.2.2(8) Followed
9.5.3(3) Followed
9.5.3(6) Followed

Where the National Annex is referenced in the clauses listed above, those National rules have been implemented.

Related Links

- [D. Eurocode 2 2004 Beam Design Brief dialog](on page 1286)
- [D. EC2 2004 Column Design Brief dialog](on page 1290)
- [D. EC2 2004 Slab Design Brief dialog](on page 1294)
- [D. UK National Annex Values dialog](on page 1300)
- [D. National Annex Values dialog](on page 1297)

D. Eurocode 2:2004 Column Design Principles

The Eurocode 2:2004 Column Design Brief is for prismatic, solid rectangular or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered. Hence only single height columns are recommended.

Design Principles

Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Reinforcement is provided by the program based on the forces generated from the analysis model and the code clauses outlined below.
The main reinforcing bars may be modified to allow for other bars passing through the section and then re-
checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which
produces the maximum required reinforcement area is called the Guiding Load Case, and the bar arrangement is
chosen based on that combination. Detailed printout of the design calculations provided is for that combination.

Code Clauses Implemented

The requirements of Eurocode 2 utilized in the design module are as follows:

2 Basis of Design

2.4.2.4 Partial factors for materials are taken from the National Annex

3 Material Properties

3.1.2.3 Implemented by the equations in table 3.1 where possible
3.1.6 Followed
3.1.7(3) Followed
3.2.3 fyk used
3.2.7(2) Horizontal top branches assumed
3.2.7(4) Followed

4 Durability and Cover to Reinforcement

4.4.1.2.(3) Followed

5 Structural Analysis

5.1.4 Followed
5.2.(5) $\theta_i$ is an input in the design brief
5.2.(7) Alternative a is followed
5.3.(7) Followed
5.8.3.1(1) Followed
5.8.3.2(1) Followed
5.8.3.3(1) Followed
5.8.4(4) Followed
5.8.8 Followed
5.8.9 Followed

6 Ultimate Limit States

6.1(6) Followed
6.2.1(2) Followed
6.2.1(3) Followed
6.2.1(4) Followed
6.2.1(5) Followed
6.2.2(1) Followed
6.2.3(1) Followed
6.2.3(2) Followed
6.2.3(3) Followed
8 Detailing of Reinforcement and Pre-stressing Tendons - general

Table 8.1N(a) Followed
8.1(2) Followed
8.2(2) Followed
8.3(2) Followed
8.5(1) Followed
8.5(2) Followed

9 Detailing of members and Particular rules

9.5.1(1) Followed
9.5.2(1) Followed
9.5.2(2) Followed
9.5.2(3) Followed
9.5.2(4) Followed
9.5.3(1) Followed
9.5.3(3) Followed
9.5.3(6) Followed

Where the National Annex is referenced in the clauses listed above, those National rules have been implemented.

Related Links

• D. Eurocode 2 2004 Beam Design Brief dialog (on page 1286)
• D. EC2 2004 Column Design Brief dialog (on page 1290)
• D. EC2 2004 Slab Design Brief dialog (on page 1294)
• D. UK National Annex Values dialog (on page 1300)
• D. National Annex Values dialog (on page 1297)

D. Eurocode 2:2004 Slab Design Principles

The Eurocode 2: 2004 Slab Design Brief is for any slab modeled with finite elements, regardless of boundary restraint conditions.

Slabs are designed for bending in two orthogonal directions for both hogging and sagging moments. Designs for shear or punching shear checks are not included.

Analysis results for plate elements that form a 'Slab' are used to generate design moments with respect to the axes of the slab. These moments are then used to generate required reinforcement contours. Note that no account is taken of the requirements for embedment or development of reinforcement.

**Tip:** The level of accuracy of results depends on the meshing density used to model the slabs. It is recommended that users examine the accuracy of the model at the analysis stage and ensure satisfactory results are obtained prior to the design stage.

Code clauses implemented are 9.3.1.1 (1) and (3) (excluding rules for concentrated loads).

Only reinforcement required to resist moment demand is designed. No other clauses are considered (no detailed layout).

Related Links

• D. Eurocode 2 2004 Beam Design Brief dialog (on page 1286)
• D. EC2 2004 Column Design Brief dialog (on page 1290)
• D. EC2 2004 Slab Design Brief dialog (on page 1294)
D. French Concrete design per BAEL

RC Designer has two BAEL design briefs, the design principles utilized by these are defined in:

**Code Ref**  B.A.E.L. 91

**Description**  Regles techniques de conception et de calcul des ouvrages et constructions en beton armé suivant la méthode des états limites.

**Revision**  Deuxième tirage 1993

D. BAEL Beam Design Principles

The BAEL 91 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to [D. Suitable Member Properties](on page 1006) for more details.

Beams are designed for flexure, and shear only. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. The user can specify the number of segments to be considered between 4 and 25 for each member.

Design for Flexure

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

a. Same size
b. Same covers

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The programs limits 8 bars in any one layer and uses a maximum of 2 layers.
The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

**Design for Shear**

The shear reinforcement is designed to resist the major axis shear force envelope, $F_z$, acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $v$, and concrete shear capacity, $v_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

![Figure 123: Minimum shear links required, V, for shear forces between these values](image)

**Anchorage and Bond Lengths**

Anchorage and bond lengths are calculated in accordance with the requirements of chapter 12. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table.

**Related Links**

- [D. BAEL 91 Beam Design Brief dialog](on page 1247)
- [D. BAEL 91 Column Design Brief dialog](on page 1252)

**D. BAEL Column Design Principles**

The BAEL 91 Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

**Design Principles**

Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the 'Guiding Load Case', and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.
The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

Related Links
- D. BAEL 91 Beam Design Brief dialog (on page 1247)
- D. BAEL 91 Column Design Brief dialog (on page 1252)

D. German Concrete design per DIN 1045-1

Concrete Design has two DIN 1045 design briefs, the design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIN 1045-1</td>
<td>Plain reinforced and prestressed concrete structures</td>
</tr>
<tr>
<td></td>
<td>Part 1: Design and construction</td>
</tr>
<tr>
<td></td>
<td>July 2001</td>
</tr>
</tbody>
</table>

Related Links
- D. DIN 1045 Beam Design Brief dialog (on page 1274)
- D. DIN 1045 Column Design Brief dialog (on page 1280)

D. DIN 1045-1 Beam Design Principles
The DIN 1045-1 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.

Beams are designed for flexure, and shear only. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. The user can specify the number of segments to be considered between 4 and 25 for each member.

Related Links
- D. DIN 1045 Beam Design Brief dialog (on page 1274)
- D. DIN 1045 Column Design Brief dialog (on page 1280)

Design for Flexure
The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

a. Same size
b. Same covers

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The programs limits 8 bars in any one layer and uses a maximum of 2 layers.
The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

Reinforcement Property

Table 29 is used for minimum reinforcement ratio.

The Stress – Strain curve is assumed as straight.

The design Fy is calculated as $F_{y}/G_{s}$ (1.15 by default)

Bending calculation for Beam

Limiting Neutral Axis depth -> clause 8.2 & 8.3

For calculating limiting moment Fig. 25 being used

Partial safety factor is calculated using clause 5.3.3- Table 2 and 5.3.3.9

Table 9 used to setup all characteristic values for normal weight concrete

Design unconfined compressive strength is calculated as in 9.1.6.2

Steel Stress

Strain in steel is calculated as

$$(d – NA) \times E_{cu} / NA$$

where

$\begin{align*}
    d & \quad \text{Effective depth} \\
    NA & \quad \text{neutral axis depth} \\
    E_{cu} & \quad \text{obtained from table 9, } E_{c2u}^{*}.001
\end{align*}$$

Stress is calculated based on Figure 26 and 27

Reinforcement Calculation

Tension reinforcement

Reinforcement required = (Force) / (Steel Stress)

Fig. 25 is used to calculate the compressive force for a balanced section.

$A_{st, \text{required.}} = C / \text{Stress in steel}$

Compression reinforcement
If compression steel is required the steel stress in compression is calculated using the strain in the steel while in compression.

\[ \text{Ast, compression} = \frac{\text{(Extra Force)}}{\text{(Stress in steel in compression)}} \]

Additional Ast needs to be added for tension = \( \frac{\text{(Extra Force)}}{\text{(Steel Stress)}} \)

Reinforcement for Axial force

For axial force reinforcement required = \( \frac{\text{(Axial Force)}}{\text{(Steel Stress)}} \)

Total Ast = Tension reinforcement + Extra Ast for compression + Ast for Axial force

Reinforcement spacing

Reinforcement spacing is calculated according to clause 12.2.2

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, \( F_z \), acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, \( v \), and concrete shear capacity, \( v_c \). From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

\[ V \]

\[ \text{zone 1 zone 2 zone 3 zone 4 zone 5 zone 6} \]

*Figure 124: Minimum shear links required, \( V \), for shear forces between these values*

Shear Reinforcement

Longitudinal reinforcement ratio is assumed as 0.02 as per 10.3.3.

\( \text{VRd,ct} \) is calculated using equation 70. (10.3.3)

\( \cot \theta \) is calculated using equation 72 (10.3.4)

\( \text{VRd,c} \) is calculated using equation 73 (10.3.4)

\[ z \text{ lever arm} = 0.9 \times \text{effective depth} \]

Minimum shear reinforcement is provided when design Shear Force < \( \text{VRd,ct} \)

Shear reinforcement is calculated using equation 75 when design shear force > \( \text{VRd,c} \)

Anchorage and Bond Lengths
Anchorage and bond lengths are calculated in accordance with the requirements of clause 3.12.8. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table.

D. DIN 1045-1 Column Design Principles
The DIN 1045 Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

Design Principles
Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the ‘Guiding Load Case’, and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

Check for slenderness
Slenderness is considered for both axis. A Column is considered if it satisfies 8.6.3.2
Critical slenderness is calculated as in 8.6.3.4

Calculation of design moment
Total eccentricity is calculated as in 8.6.5.5
e2, eccentricity is calculated as in 8.6.5.7

ea, the additional accidental eccentricity is calculated as in 8.6.4
The minimum eccentricity is calculated as in 8.6.3.9
For slender column e0, design first order eccentricity is in accordance with 8.6.5.6

Reinforcement limits
Maximum % of steel is calculated in accordance with 13.5.2.2
Minimum % of steel is calculated in accordance with 13.5.2.1

The shear reinforcement design in column is calculated in accordance with 10.3
The link spacing is in accordance with 13.5.3.4

Related Links
- D. DIN 1045 Beam Design Brief dialog (on page 1274)
- D. DIN 1045 Column Design Brief dialog (on page 1280)

D. Indian Concrete design per IS456
Concrete Design has two IS 456 design briefs, the design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>IS 456</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>Plain and Reinforced Concrete - Code of Practice</td>
</tr>
</tbody>
</table>
D. IS456 General Considerations
This section describes some of the material properties, section properties, and other assumptions made for the
design of concrete beams and columns per IS456:2000 in RC Designer.

Materials
The following clauses from section 2 of IS 456:2000 are considered by the program:

- Cl 5.3.3 – The size of aggregate is considered as 20 mm.
- Cl. 5.6 – This implementation assumes that the reinforcement specified conforms to Cl. 5.6 of the code. The
  modulus of elasticity of the reinforcement is taken as 200,000 N/mm$^2$ as per clause 5.6.3 of the code.
- Cl. 6.2 – The program will only consider the design of elements that use ordinary concrete or Standard
  concrete.
  If you specify a high strength concrete grade (i.e., compressive strength greater than 55 N/mm$^2$), the
  program will issue a warning alert you that there might be additional considerations that need to be taken
  into account in the design. The design strengths of concrete shall be as given in Table 2 of IS 456:2000.
- Cl 6.2.2 – The tensile and flexural strength of concrete shall be taken as:
  \[
  f_{cr} = 0.7 \cdot \sqrt{f_{ck}}
  \]
  where:
  \[
  f_{ck} = \text{the characteristic compressive strength of concrete.}
  \]
- Cl. 6.2.3.1 – The modulus of elasticity of concrete is taken:
  \[
  E_c = 5000 \cdot \sqrt{f_{ck}}
  \]

Note: The effects of creep, shrinkage and thermal expansion are ignored.

General
This design is based on Limit state method. The Working Stress method in Annex B of IS 456:2000 is not
considered for this implementation. The loads and forces considered are as per Clause 19 and are defined in
STAAD.Pro. The density of concrete is taken as 24 KN/m$^3$. The stability of the structure as a whole (i.e.,
resistance to overturning, sliding, or lateral sway) are not considered by the program for the design of these
elements. This implementation assumes that the section dimensions comply with fire resistance according to
Clause 21 of the code.

The following clauses from section 3 of IS 456:2000 are considered by the program:

- Cl. 22.2 – The effective span for beams used is as defined in Cl. 22.2 (a) – (d) of the code. The clear span is
  taken as the node-to-node distance of a member (end nodes or supported nodes) in the STAAD.Pro model.
  Note: You will have the option to edit the effective length in the column (on page 1328) design brief.
- Cl. 22.3 – The relative stiffness for the members is based on the gross cross section where necessary. The
  design of the member is based on its gross cross section.
- Cl. 22.6.2 – The critical section for shear is considered to be at the face of the support. Enhanced shear at
  sections up to a distance equal to the effective depth of the beam are also considered.
- Cl. 23 – Refer to D. IS456 Beam Design Principles (on page 1057) for details of beam design.
- Cl. 25 – Refer to D. IS456 Column Design Principles (on page 1059) for details on column design
- Cl. 26 – Requirements for reinforcement and detailing:
Cl. 26.1.1 – The reinforcement provided in the design of beams is limited to four layers (i.e., two top & two bottom) for this implementation. Bundling of reinforcement will not be considered in this implementation.

26.1.2 – Consideration will also be given to the detailing rules as per IS 13920.

26.2.1 – The development length of bars is taken as:

\[ L_d = \frac{\varphi \cdot \sigma_s}{(4 \cdot \tau_{bd})} \]

where:

\( \varphi \) = nominal diameter of the bar
\( \sigma_s \) = stress in bar at the section considered at design load
\( \tau_{bd} \) = design bond stress given in 26.2.1.1

**Note:** Clause 26.2.1.1 gives the bond stress for bars in tension in the table. For bars in compression these values need to be increased by 25%. The anchorage length of straight bars in compression shall be in accordance with 26.2.1.

26.2.2.5 – The bearing stress at bends is considered and is evaluated as:

\[ \text{Bearing Stress} = \frac{F_{bt}}{(r \cdot \varphi)} \]

where:

\( F_{bt} \) = tensile force due to design loads in a bar or group of bars
\( r \) = internal radius of the bend
\( \varphi \) = size of bar or, in a bundle, the size of bar of equivalent area

The bearing stress within a bend or a hook should not exceed 1.5 \( F_{ck} / (1 + 2\varphi /a) \), where \( a \) is the center to center distance between bars.

Cl.26.2.3 - The curtailment of reinforcement shall be done as per the conditions given in cl. 26.2.3.1 to 26.2.3.4 of the code.

Cl.26.3 – The spacing of reinforcement shall be done as given in Cl.26.3.1 – 26.3.3 of the code.

Cl. 26.4 – The cover to reinforcement is specified by the user. However checks is done to make sure that the cover requirements as per Cl. 26.4.1 – 26.4.3 are met.

Cl.26.5 – The minimum reinforcement requirements as given in Cl. 26.5.1.1 is checked for beams and columns. Separate checks is done for tension, compression and side face reinforcements. Transverse reinforcements for beams will also be checked as per Cl. 26.5.1.4 - 26.5.1.8 of the code. The reinforcement requirements for columns is checked as per Cl. 26.5.3 of the code.

### Related Links
- D. IS456 Beam Design Brief dialog (on page 1325)
- D. IS456 Column Design Brief dialog (on page 1328)

**D. IS456 Beam Design Principles**

The IS456 Beam Design Brief is for single or multi span prismatic, rectangular members. The member sections must be setup in the Section Property Calculator and assigned from the User section database.

**Note:** Refer to the 'Section Property Calculator' manual for setting up standard sections and User databases.

Beams are designed for flexure and shear. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. You may specify the number of segments to be considered between 4 and 25 for each member.
Both Ultimate limit state and Serviceability Limit states will be considered. Serviceability Limit states will be taken into account by satisfying the span/effective depth ratios specified in Cl. 23.2.1 of the code.

Partial Safety factors

The partial safety factors for materials used for member design are as follows:

\[ \Gamma_m \text{ for concrete} = 1.5 \]
\[ \Gamma_m \text{ for steel} = 1.15 \]

Design for Flexure

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Lateral bending are considered if this option is selected. Compression reinforcement is provided where required.

The design for flexure is performed as per Cl. 38 of the code.

The design of a beam is based on an envelope of design forces. Thus, at each of the defined sections the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams. The sub-beams which can use the same reinforcing cage are those with:

a. Same size
b. Same covers

For each sub-beam, the sections with the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The program is limited to eight bars in any one layer and uses a maximum of two layers.

The program then checks each section along the beam to determine how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, \( F_z \), acting through the beam. The minor axis shear and tensional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore, the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress \( \tau \) and concrete shear capacity \( \tau_c \). From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

The design for shear reinforcement is performed as per Cl.40 of the code. The nominal shear stress in beams is evaluated as:

\[ \tau = \frac{V}{bd} \]

where:

\[ V = \text{the shear force} \]
\[ b = \text{the width of the section taking the shear force} \]
\[ d = \text{the effective depth} \]
The design shear strength of concrete is taken as specified in Cl. 40.2 of the code. For members in axial compression, the design shear strength is multiplied by the following factor:

$$\delta = 1 + \frac{(3 \, P_u)}{(A_g \times f_{ck})}$$

where

- $P_u$ = the axial compressive load
- $A_g$ = the gross concrete area
- $f_{ck}$ = the characteristic compressive strength

Design for shear reinforcement is based on Cl.40.3 to 40.5 of the code. Note that the design brief includes the option of selecting whether to consider enhanced shear effects near the supports of the beam. If the option to consider enhanced shear is selected, the program also considers Cl.22.6.2 when working out the shear links.

Design for Torsion

The design of beams considers the combined effect of shear and torsion per Cl. B-6.3 of IS 456-2000. The design for torsion is based on Cl. 41.3 of IS 456. This method involves working out an equivalent bending moment and equivalent shear based on the torsional moment at a particular section. The equivalent shear force for torsion design is evaluated as:

$$V_e = V_u + 1.6 \frac{T_u}{b},$$

where

- $V_u$ = shear force at the section
- $T_u$ = torsional moment
- $b$ = the breadth of the beam

The longitudinal reinforcement required at a particular section subject to torsion is evaluated based on an equivalent bending moment evaluated as:

$$M_e = M_u + M_t$$

where

- $M_u$ = the bending moment at the cross section
- $M_t = T_u \{1+(D/b)\} / 1.7 \}$

The distribution of torsion reinforcement will then be done as per Cl. 26.5.1.7 of IS456.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore, the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress $\nu$ and concrete shear capacity, $\nu_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

Anchorage and Bond Lengths

Anchorage and bond lengths are calculated in accordance with the requirements of Cl. 26.2 of IS456:2000. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table and export to Multi-RC.

Related Links

- [D. IS456 Beam Design Brief dialog](on page 1325)
- [D. IS456 Column Design Brief dialog](on page 1328)

D. IS456 Column Design Principles

The IS456 Column Design Brief is for prismatic, rectangular solid, or circular shaped members.
As with Beam Design (on page 1057), section properties must be taken from the User databases. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

**Note:** Refer to the Section Property Calculator manual for setting up standard sections and User databases.

**Principles**

Columns are designed for axial force and biaxial end moments, Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

If the member has a reinforcement cage already defined, then the program checks the reinforcement for compliance. Otherwise the program will select a reinforcement cage from the bars and specifications of the Design Brief to minimize the area of main reinforcement bars.

**Tip:** Selecting a column member from the design group and selecting **Edit > Copy**, copies the reinforcement cage which can be pasted on columns in the Design Group of the same dimensions, which can then be checked with that same reinforcement.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the Guiding Load Case, and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The program conforms to the following requirements of section 39 of IS456: 2000 in the design module:

<table>
<thead>
<tr>
<th>Clause</th>
<th>Content</th>
</tr>
</thead>
<tbody>
<tr>
<td>39.2</td>
<td>Minimum Eccentricity</td>
</tr>
<tr>
<td>39.3</td>
<td>Short Axially Loaded member in compression</td>
</tr>
<tr>
<td>39.5</td>
<td>Members Subjected to Combined Axial Load and Uniaxial Bending.</td>
</tr>
<tr>
<td>39.6</td>
<td>Members Subjected to Combined Axial Load and Biaxial Bending.</td>
</tr>
<tr>
<td>39.7</td>
<td>Slender Compression Member</td>
</tr>
</tbody>
</table>

**Note:** Compression members with helical reinforcement (CL 39.4) are not considered by the program.

The design for compression members is based on Cl. 25 of IS456:2000. In addition to the assumptions made for beam design, the following assumptions are made for compression design:

- The maximum compressive strain in concrete in compression is 0.002
- When there is no tension in the concrete section and the section is subject to bending and axial compression, the maximum compressive strain at the highly compressed extreme fiber is taken as 0.0035 – 0.75 x strain in the least compressed extreme fiber.

Refer to the design brief (on page 1328) for an explanation of the input parameters.
All compression members are designed for a minimum eccentricity in accordance with Cl. 25.4 of IS456. Compression members are treated as either ‘short’ or ‘slender’ in accordance with Cl. 25.1.2 of IS456. A compression member will be considered as short when the slenderness ratios:

\[
l_{ex}/D < 12 \quad \text{and} \quad l_{ey}/b < 12
\]

where

\begin{align*}
l_{ex} &= \text{the effective length with respect to the major axis} \\
l_{ey} &= \text{the effective length with respect to the minor axis}
\end{align*}

If either of the slenderness ratios exceeds 12, the compression member is considered to be slender. The axial load capacity of a “short” compression member is calculated as per Cl. 39.3 of IS456. Compression members subject to combined axial load and bending must also satisfy the interaction equation as per Cl. 39.6 of IS456:2000. In the case of slender compression members, any additional moments are automatically calculated and used as per Cl. 39.7 of IS456. The correction factor ‘k’ used is also automatically calculated as per Cl. 39.7.1.1 of IS456.

The design procedure used involves an iterative process where the neutral axis position for the design section is evaluated for a given combination of axial force and/or bending moment. The bar arrangement is chosen based on the Guiding Load Case. The program systematically goes through all the bar sizes given in the brief to evaluate a symmetrical arrangement of bars on all four sides of the section. For each of the design sections thus considered, the program attempts to evaluate the neutral axis position for the given set of design forces. The program considers the neutral axis position from being at the extreme compressive fibers (edge of the section with the highest compressive force) up to a distance equal to four times the section depth. If the neutral axis depth is greater than 4 x D, the program classifies the chosen section as failed. Once the exact position on the neutral axis is determined, the program then evaluates the section moment capacities for the given axial load, about both axes. Subsequent interaction checks are also performed if necessary. Among all the design sections that are deemed satisfactory for the given set of forces, the program automatically selects the most efficient bar arrangement and reports the details. Upon determining a suitable main bar arrangement, the program then evaluates the required shear reinforcement for the section.

Related Links

- D. IS456 Beam Design Brief dialog (on page 1325)
- D. IS456 Column Design Brief dialog (on page 1328)

D. IS 13920 Seismic Design

Seismic checks per IS 13920:1993 can be performed with the concrete design module in STAAD.Pro (RC Designer). A valid IS456 beam and/or column design must be performed first in order to proceed with the seismic checks as per IS13920.

The following clauses are dealt with in the current version of the RC Designer:

Checks for beam members

In general, the following clauses from IS 13920 have been considered for beam design:

- Cl. 6.1.1 – The member will be checked to ensure that axial stress does not exceed 0.1 x fck (concrete strength)
- Cl. 6.1.2 & Cl. 6.1.3 – The program will issue a warning if the dimensions do not satisfy the limits in the code as per the clauses above.
- Cl. 6.1.4 – The program will issue a warning if the depth of the member exceeds ¼ of the span of the member.
- Cl. 6.2.1 & Cl. 6.2.2 – The longitudinal reinforcement ratios shall be checked as per the limits specified in the clauses above.
- Cl. 6.3 – The shear reinforcement shall be checked for a design shear force as specified in Cl. 6.3. of IS13920.

Checks for column members
In general, the following clauses from IS 13920 have been considered for column design:

Cl. 7.1.1 – The member will be checked to ensure that axial stress does not exceed 0.1 x fck (concrete strength)
Cl. 7.1.2 & 7.1.3 – The program will issue a warning if the dimensions do not satisfy the limits in the code as per the clauses above.
Cl. 7.3 – The program will work out the critical shear force as defined in Cl. 7.3.4 of the code.
Cl. 7.4 – The confining reinforcement (shear links) for the critical zones will be checked as per Cl. 7.4 of the code. The program will issue a pass/fail status for the shear links specified within the critical zones of the column. The Earthquake Reinforcement page (on page 1216) will indicate the status as 'pass' if the shear link spacing criteria of IS13920 are satisfied within the critical zones. Otherwise the program will indicate a 'fail' status.

Related Links
- D. Designing longitudinal and shear reinforcement in beams per IS13920 (on page 1062)

D. Designing longitudinal and shear reinforcement in beams per IS13920

1. In the IS456 Beam Design Brief dialog General tab (on page 1325), set the Check for IS13920 option.
2. Perform a beam design.
3. Select the Earthquake page (on page 1216).
4. Select the appropriate load case is selected to perform the seismic checks to allow for a strong column-weak beam mechanism.

The program internally calculates the moment capacities of the beams and columns framing into a joint and assign a 'pass/fail' status depending on the corresponding beam and column capacities at that joint for the direction under consideration. If the sum of the column capacities is greater than the sum of the beam capacities in a particular direction, the program will assign a 'pass' status. If on the other hand, the sum of the column capacities is greater than the beam capacities, the program will assign a 'fail' status. The program will also check for the shear reinforcement capacities and shear link spacings in the "critical zones" as defined in IS13920.

5. Select the Earthquake Reinforcement page to review the status of the IS 13920 design.

Related Links
- D. IS 13920 Seismic Design (on page 1061)

D. Japanese Concrete design per AIJ

RC Designer has one AIJ design brief, the design principles utilized by this is defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>Description</th>
<th>Revision</th>
</tr>
</thead>
<tbody>
<tr>
<td>AIJ</td>
<td>Standard for structural calculation of reinforced concrete structures</td>
<td>1985</td>
</tr>
</tbody>
</table>

D. AIJ Beam Design Principles
The AIJ Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.
Beams are designed for flexure, and shear only. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. The user can specify the number of segments to be considered between 4 and 25 for each member.

Design for Flexure

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

**a.** Same size  
**b.** Same covers

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The programs limits 8 bars in any one layer and uses a maximum of 2 layers.

The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

Design procedure

**Fy** – allowable unit stress for reinforcement bars is calculated based on Table 5 and considers both temporary and permanent loading

2. Calculate Neutral axis ratio based on Art. 14, equation 8.

Allowable bending moment = \(Cbd^2\)

Gamma is considered to be 0.4 to begin with for light weight concrete.

The area of reinforcement, \(A_t\), is calculated using Art. 14.3, equation 14 when the allowable bending moment is greater than the design moment.

An iterative method is used to calculate area of reinforcement required when design moment is greater than allowable moment.
1. First program tries to increment compression reinforcement by incrementing gamma.
2. If gamma reaches 1.0, then program increments tension reinforcement and recalculates “C”.
3. The design fails if the tension reinforcement exceeds maximum reinforcement.

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, Fz, acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, \( v \), and concrete shear capacity, \( v_c \). From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

![Figure 125: Minimum shear links required, V, for shear forces between these values](image)

Shear capacity is calculated in accordance with Art. 16.2.1, equation 22

\[ f_s \] is calculated in accordance with table 4.

To calculate ‘pw’, spacing is first assumed to be 45 cm. and the program runs through a iterative method and reduces the spacing to establish the actual required spacing.

The design checks for minimum and maximum spacing as per Art.16.4.ii

Anchorage and Bond Lengths

Anchorage and bond lengths are calculated in accordance with the requirements of chapter 12. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table.

Related Links

- D. AIJ Beam Brief dialog (on page 1239)

D. Norwegian Concrete design per NS3473

Concrete Design has two NS3473 design briefs, the design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NS 3473</td>
<td>Concrete Structures</td>
</tr>
<tr>
<td></td>
<td>Design Rules</td>
</tr>
</tbody>
</table>
The NS 3473 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISOMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.

Beams are designed for flexure, and shear only. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. The user can specify the number of segments to be considered between 4 and 25 for each member.

Design for Flexure

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

- a. Same size
- b. Same covers

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The programs limits 8 bars in any one layer and uses a maximum of 2 layers.

The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, Fz, acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $v$, and concrete shear capacity, $v_c$. From this, the section is classified as either minimum link or a high shear section.
Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

![Diagram showing shear forces between zones](image)

*Figure 126: Minimum shear links required, V, for shear forces between these values*

Anchorage and Bond Lengths

Anchorage and bond lengths are calculated in accordance with the requirements of Eurocode 2. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table and export to Multi-RC.

**Related Links**
- [D. NS 3473 Beam Design Brief dialog](on page 1331)
- [D. NS 3473 Column Design Brief dialog](on page 1337)

D. NS3473 Column Design Principles

The NS 3473 Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

**Design Principles**

Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the 'Guiding Load Case', and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

**Related Links**
- [D. NS 3473 Beam Design Brief dialog](on page 1331)
- [D. NS 3473 Column Design Brief dialog](on page 1337)

**D. Russian Concrete design per SP 52-101-03**

RC Designer has two SP 53-101-03 design briefs, the design principles utilized by these are defined in:
Design
D. Concrete Design

Code Ref  SP 52-101-2003
Description "Concrete And Reinforced Concrete Structures Without Prestressing," Concrete and Reinforced Concrete Research and Technological Institute (НИИЖБ)
Revision  2006

D. SP52-101-03 Beam Design Principles
Design of the multi-span reinforced concrete beam is performed according to the Russian design code SP52-101-2003 Concrete And Reinforced Concrete Structures Without Prestressing, 2004. The program computes required reinforcement areas and performs necessary RC detailing operations. The ultimate limit state and serviceability limit state verification as well as detailing requirements of the SP52-101-2003 are provided in this program.

Present Russian RC design code, SP52-101-2003 Concrete And Reinforced Concrete Structures Without Prestressing, coincides with 1990 CEB – FIB Model Code for Concrete Structures in the part of normal section analysis. However, shear and torsion analysis and cracking calculations are based on rather different principles.

Design for Bending
Analytical model for planar bending is based on the following figure:

![Figure 127: Planar bending of RC section](image)

where

- \( R_b \) = design strength of concrete
- \( R_s \) = design strength of steel
- \( x \) = depth of neutral axis, constrained by the limiting quantity, depending on steel strength

In the program the small value of the axial force can be taken into account, provided the compression force does not cause the neutral axis depth greater than the limiting value, and in the tension case, the hog zone remains compressed.
The bending strength of the section and the layout of the longitudinal reinforcing steel are interrelated through the effective depth of the section. The crack width limitation must also be taken into account when choosing the longitudinal reinforcement. On the other hand, one must try all possible load combinations having in mind the interaction of the axial force and bending moment.

Strength and crack width verification is performed for the set of design sections uniformly spaced in a span. The user can introduce additional checking points for each span where he presumes the existence of the sharp maximum of the moment caused by the concentrated force. The critical load combination is defined by the maximum value of the required tension steel area for each design section. The sag- and hog-critical sections are defined by the maximum of required tension reinforcement. The layout of tension steel bars is defined for both sag- and hog-critical section of the beam.

All admissible bar diameters are tried during design of critical section; the diameter resulting in minimum steel area is adopted as optimal one. As the result, the optimum layout for critical sag reinforcement and critical hog reinforcement is obtained.

The tension bar layouts are produced only for critical sections. For other design sections, the bar layouts are obtained during bar curtailment procedure as illustrated in the following figure. Required steel areas, computed previously, are used during this procedure.

The use of this design method is restricted to beams with predominantly downward-directed loads, when maximum sag moments are located in about the middle of span and maximum hog moments (if any) are at the supports. In this case the provided steel areas at all design sections are straightforwardly obtained in the sequential bars curtailment procedure.
Design for Shear

It is assumed that the shear resistance of member consists of the following two factors:

- concrete compression zone shear $Q_b$
- contribution of links $Q_{SW}$

The contribution of inclined reinforcing steel is not considered.

![Figure 129: Shearing of RC member](image)

The following formula is used to calculate crack width:

$$a_{cr} = \varphi_1 \varphi_2 \psi_s (\sigma_s / E_s) l_s$$

where

- $\sigma_s$ = steel stress computed according to elastic formulas, excluding concrete tension zone
- $l_s$ = spacing of cracks
- $\varphi_1$ = 1.0 for short term loading, = 1.4 for long term loading
- $\varphi_2$ = 0.5 for high bond bars, = 0.8 for bars with plain surface
- $\psi_s$ = factor allowing for steel-concrete interaction in the tension zone $\leq 1.0$,

The spacing of cracks is calculated from the expression

$$l_s = 0.5(A_{bt}/A_S) d_s$$

where

- $A_{bt}$ = concrete tension area
- $A_S$ = steel tension area
- $d_s$ = diameter of reinforcing bars

The torsional strength includes the contributions of both the longitudinal and transverse reinforcement.

The longitudinal reinforcement is checked according to strength and crack width conditions. The transverse bars must comply with the inclined sections strength and maximum allowable spacing conditions. Additionally, in the presence of torsion moments the combined action of bending, shear and torsion must be taken into account.

Design Procedure

The design process is summarized by the following steps:
1. Find the sag- and hog-critical sections of the continuous beam, according to the maximum required tension steel area. Typically, these sections are situated at a support and at about the middle of a span.

2. Produce the layouts of tension reinforcing steel for these two critical sections.

3. Produce the derived layouts for all supports and most adverse middle span sections of each beam.

4. Carry out the reinforcing steel curtailment procedure for each span of the continuous beam.

5. Compute the required stirrup spacing for each section of the continuous beam. In the detailing procedure, the stirrup spacing may be manually defined for each part of the span.

6. The final strength check is performed for the obtained layout for each design section.

The assumptions adopted for the beam design procedure, can be summarized as follows:

1. The diameters of bottom and top reinforcing bars may be different; all bars in the bottom or top of the continuous beam are of the same diameter.
2. The sag- and hog-critical sections of the continuous beam are selected according to the most adverse load conditions; basic reinforcement layouts are produced for these two critical sections.
3. The diameters of the bars for the sag- and hog-critical reinforcement are chosen according to the minimum steel area criterion, within the reasonable boundaries you have imposed.
4. The tension bars compatibility is retained at support sections of the continuous beam. The sag bars layouts and diameters in each span can be different.
5. In each span of the continuous beam the derived layouts, compatible with the basic layouts, are to be produced.
6. Derived layouts are based on the same reinforcing bar diameters as basic ones.
7. Bar curtailment is based on the loading and strength conditions of the set of the beam cross-sections; these cross-sections must not coincide with definite nodes of the finite element mesh.

Related Links

- D. SP 52 101-03 Beam Brief dialog (on page 1342)
- D. SP 52 101-03 Column Brief dialog (on page 1344)

Eccentrically Loaded Columns

The strength of the section and the layout of the longitudinal reinforcing steel bars are interrelated through the effective depth of the section. The program evaluates all selected load combinations in the envelope to determine the worst case interaction of the axial force and bending moment.

Strength verification is performed for the section with maximum axial force eccentricity. The critical load combination is defined by the maximum value of the required steel area. All admissible bar diameters are tried during design of critical section. The diameter resulting in minimum steel area is adopted as optimal one. The transverse reinforcement is determined using maximum shear force value.
**Design Procedure**

The design process is summarized by the following steps:

1. For each load combination, find the critical section of the column, according to the maximum axial force eccentricity.
2. Design the reinforcing steel for the critical section.
3. Compute the required stirrup spacing for the column.
4. Perform the final strength check for the obtained longitudinal and transverse reinforcement.

The assumptions adopted for the column design procedure, can be summarized as follows:

1. The longitudinal reinforcement of the column is symmetrical.
2. Diameter of the intermediate longitudinal bars in the section can be less than that of the corner bars.

**Related Links**

- [D. SP 52 101-03 Beam Brief dialog](on page 1342)
- [D. SP 52 101-03 Column Brief dialog](on page 1344)

**D. Singaporean Concrete design per CP65**

Concrete Design has two CP 65 design briefs, the design principles utilized by these are defined in:
D. CP65 Beam Design Principles

The CP 65 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.

Beams are designed for flexure, and shear only. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. The user can specify the number of segments to be considered between 4 and 25 for each member.

Design for Flexure

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

- Same size
- Same covers

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The programs limits 8 bars in any one layer and uses a maximum of 2 layers.

The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.
Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, $F_z$, acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $\nu$, and concrete shear capacity, $\nu_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

![Figure 131: Minimum shear links required, V, for shear forces between these values](image)

Anchorage and Bond Lengths

Anchorage and bond lengths are calculated in accordance with the requirements of clause 3.12.8. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table.

Code Clauses Implemented

The following lists the code clauses used from CP65-1999.

3.0 Design and detailing: Reinforced concrete

3.1.7 Strength of materials controlled by brief

3.3 Concrete cover to reinforcement controlled by brief

3.4.4.1 Analysis of section Program Conforms

3.4.4.4 Design formulae for rectangular beams. Program Conforms

3.4.4.5 Design formulae for flanged beams where the neutral axis falls below the flange. Program Conforms

3.4.5 Design shear resistance of beams

3.4.5.2 Shear stress in beams. Program Conforms

3.4.5.3 Shear reinforcement: form, area and stress. Program Conforms

3.4.5.4 Concrete shear stresses. Program Conforms

3.4.5.5 Spacing of links. Program Conforms

3.4.5.10 Enhanced shear strength near supports. Program Conforms

3.4.5.12 Shear and axial load. Program Conforms
3.4.6 Deflection of beams

3.4.6.3 Span/effective depth ratio for a rectangular or flanged beam. Program Conforms
3.4.6.4 Long spans. Program Conforms
3.4.6.5 Modification of span/depth ratios for tension reinforcement. Program Conforms
3.4.6.6 Modification of span/depth ratios for compression reinforcement. Program Conforms
3.4.12.3 Minimum percentage of reinforcement. Program Conforms
3.4.12.6.1 Maximum percentage of reinforcement. Program Conforms
3.12.7.2 Arrangement of links for containment of beam or column compression. Program Conforms

3.12.11 Spacing of reinforcement

3.12.11.1 Minimum distance between bars. Program Conforms
3.12.11.2 Maximum distance between bars in tension. Program Conforms
3.12.11.4 Clear distance between bars in tension. Program Conforms
3.12.11.2.5 Clear distance between face of beam and nearest longitudinal bar in tension. Program Conforms
3.12.11.2.6 Bars near side faces of beams exceeding 750 mm overall depth. Program Conforms

Related Links

- D. CP65 Beam Design Brief dialog (on page 1269)
- D. CP65 Column Design Brief dialog (on page 1271)

D. CP65 Column Design Principles

The CP 65 Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

Design Principles

Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the 'Guiding Load Case', and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

Code Clauses Implemented

The requirements of section 3.8 of CP65: Part 1 1999 utilized in the design module are as follows:

3.8 Column

3.8.1 General. Program Conforms
3.8.1.1 Symbols. Program Conforms
3.8.1.2 Size of columns. N/A
3.8.1.3 Short and slender columns. Program Conforms
3.8.1.4 Plain concrete columns. N/A
3.8.1.5 Braced and unbraced columns. Program Conforms

3.8.1.6 Effective height of a column
   3.8.1.6.1 General. Program Conforms with Beta as a user input
   3.8.1.6.2 End conditions. Program Conforms with Beta as a user input

3.8.1.7 Slenderness limits for columns. Program conforms
3.8.1.8 Slenderness of unbraced columns. Program conforms

3.8.2 Moments and forces in columns
   3.8.2.1 Columns in monolithic frames designed to resist lateral forces. User decision
   3.8.2.2 Additional forces induced by deflection at ULS. Program Conforms for column being designed, not for
       members connected
   3.8.2.3 Columns in column and beam construction or in monolithic braced structural frames. Forces to be
       designed for are determined by the users own model.
   3.8.2.4 Minimum eccentricity. Program conforms

3.8.3 Deflection induced moments in solid slender columns
   3.8.3.1 Design
   3.8.3.2 Design moments in braced columns bent about a single axis. Program Conforms
   3.8.3.3 Slender columns bend about a single axis( major or minor). Program Conforms
   3.8.3.4 Columns where le/h exceeds 29, bent about their major axis. Program Conforms
   3.8.3.5 Columns bent about their major axis. Program Conforms
   3.8.3.6 Slender columns bent about both both axes. Program Conforms
   3.8.3.7 Unbraced structures. Program Conforms
   3.8.3.8 Deflection of unbraced column. N/A
   3.8.3.9 Additional moments on members attached to a slender column. N/A

3.8.4 Design of column section for ULS
   3.8.4.1 Analysis of sections. Program Conforms
   3.8.4.2 Design charts for symmetrically-reinforced columns. N/A
   3.8.4.3 Nominal eccentricity of short columns resisting moments and axial loads. N/A
   3.8.4.4 Short braced columns supporting an approximately symmetrical arrangement of beams. N/A
   3.8.4.5 Biaxial bending. Program Conforms
   3.8.4.6 Shear in columns. Program Conforms
   3.8.5 Deflection of columns. N/A
   3.8.6 Crack control in columns. The clause states that “if crack check is required, then the member should be
       checked as a beam”

N/A is to be interpreted as either not application or not implemented.

Clauses that do not appear in the list are deemed not to be considered by the program.

**Related Links**
- [D. CP65 Beam Design Brief dialog](on page 1269)
- [D. CP65 Column Design Brief dialog](on page 1271)
Design

D. Concrete Design per EHE

Concrete Design has two EHE design briefs, the design principles utilized by these are defined in:

**Code Ref**  EHE  
**Description**  Instruccion de Hormigon Estructural. Con comentarios de los miembros de la Comision Permanente del Hormigon.  

D. EHE Beam Design Principles

The EHE Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.

Beams are designed for flexure, and shear only. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. The user can specify the number of segments to be considered between 4 and 25 for each member.

Design for Flexure

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

a. Same size

b. Same covers

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The programs limits 8 bars in any one layer and uses a maximum of 2 layers.

The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

Design for Shear
The shear reinforcement is designed to resist the major axis shear force envelope, $F_z$, acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $v$, and concrete shear capacity, $v_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

![Figure 132: Minimum shear links required, V, for shear forces between these values](image)

Anchorage and Bond Lengths

Anchorage and bond lengths are calculated in accordance with the requirements of clause 3.12.8. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table.

**Related Links**

- [D. EHE Beam Brief](on page 1307)
- [D. EHE Column Design Brief dialog](on page 1313)

**D. EHE Column Design Principles**

The EHE Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

**Design Principles**

Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the “Guiding Load Case”, and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

**Related Links**

- [D. EHE Beam Brief](on page 1307)
D. Turkish Concrete design per TS 500

Concrete Design has two TS 500 design briefs, the design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>TS 500</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>Building code requirements for reinforced concrete by the Turkish 'TS 500' code.</td>
</tr>
<tr>
<td>Revision</td>
<td>February 2000</td>
</tr>
</tbody>
</table>

**Note:** The TS-500 code is very much influenced by the ACI-318 design code; many of the formulas and requirements are exactly the same, with certain differences in constants and properties.

D. TS 500 Beam Design Principles

The TS 500 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to [D. Suitable Member Properties](on page 1006) for more details.

Beams are designed for flexure, and shear only. Each member is divided into equally spaced sections and the locations of maximum positive and negative moments along each element that makes up the member. The user can specify the number of segments to be considered between 4 and 25 for each member.

**Design for Flexure**

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

a. Same size
b. Same covers

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The programs limits 8 bars in any one layer and uses a maximum of 2 layers.
The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, \( F_z \), acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, \( v \), and concrete shear capacity, \( v_c \). From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

![Diagram of shear links](image)

**Figure 133: Minimum shear links required, \( V \), for shear forces between these values**

Design for Torsion

If the option to 'Include Torsion Effects' is selected in the Beam Design Brief - General. Then the design is modified by using:

\[
V_{cr} = 0.65f_{cd}b_w d \\
T_{cr} = 1.35f_{cd}T_S
\]

(Eqn. 8.11)

where

- \( T_{cr} \) = torsional moment that causes torsional cracking
- \( S \) = sectional constant

TS-500 uses a simplified \( S \) in the calculations.

\[
\psi = \left( \frac{V_d}{V_{cr}} \right)^2 + \left( \frac{T_d}{T_{cr}} \right)^2
\]

where

- \( V_d \) and \( T_d \) = the computed shear and torsional values
- \( > V_{cr} \) and \( > T_{cr} \) = are the cracked section shear and torsion values

If this is less than 1.0 then the section is classed as 'not cracked', but minimum reinforcement should be provided.

If it is greater or equal to 1.0 then it is classed as cracked and the torsional stress must be checked. If this stress is greater than 0.25 \( f_{cd} \), then the cross section is inadequate and the section must be increased.
S is calculated according to its shape thus:

<table>
<thead>
<tr>
<th>Shape</th>
<th>Section Constant</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rectangular</td>
<td>( b_w \frac{y}{h} )</td>
</tr>
<tr>
<td>Tee</td>
<td>( \Sigma \left( x^2 \frac{y}{h} \right) )</td>
</tr>
<tr>
<td>Circular</td>
<td>( \pi \frac{d^3}{12} )</td>
</tr>
</tbody>
</table>

where 'x' and 'y' = the short and long edges respectively for the rectangles that form the Tee section.

The maximum Torsional stress is calculated from:

\[ \tau_{\text{max}} = 0.22 f_{\text{cd}} \]

The actual torsional stress is calculated as:

\[ \tau = \tau_{s0} + \tau_{to} \]

or

\[ \tau = \frac{V_d}{b_w d} + \frac{T_d}{S} \]

which must be less than or equal to 0.22 \( f_{\text{cd}} \) (equation 8.19).

The recommendation for torsional reinforcement is:

a. Longitudinal torsional reinforcement should be placed around the section perimeter and must be at least 12 mm diameter, with the maximum spacing of 30 cm.

b. Link bars should be at least 8 mm diameter.

c. The maximum link spacing should be the minimum of \( d/2 \), \( ue/8 \) and 30 cm.

d. Torsion requires greater anchorage. Use minimum 10 diameter length as free end for hook

Anchorage and Bond Lengths

Anchorage and bond lengths are calculated in accordance with the requirements of chapter 12. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table.

Related Links

- D. TS500 Beam Design Brief dialog (on page 1347)
- D. TS500 Column Design Brief dialog (on page 1350)

D. TS 500 Column Design Principles

The TS 500 Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

Design Principles
Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the 'Guiding Load Case', and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

**Related Links**
- *D. TS500 Beam Design Brief dialog* (on page 1347)
- *D. TS500 Column Design Brief dialog* (on page 1350)

### D. British Concrete design per BS 8110

Concrete Design has three BS8110 design briefs, the design principles utilized by these are defined in:

**Table 1**

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>BS 8110</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>Structural Use of Concrete</td>
</tr>
<tr>
<td>Part 1. Code of practice for design and construction</td>
<td></td>
</tr>
<tr>
<td>Part 2. Code of practice for special circumstances (used for torsion design)</td>
<td></td>
</tr>
<tr>
<td>Revision</td>
<td>Part 1:1997</td>
</tr>
<tr>
<td></td>
<td>Part 2:1985 (used for torsion design)</td>
</tr>
</tbody>
</table>

**D. BS8110 Beam Design Principles**

The BS8110 Beam Design Brief is for single or multi span, prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to *D. Suitable Member Properties* (on page 1006) for more details.

Beams are designed for bending, shear and deflection. They may also be checked for Torsion. Sections are taken at equal increments along each span of the beam and at the positions of maxima of hogging and sagging moments and maximum positive and negative shear.

**Design for Bending**

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment at that section.

The beam is then divided into sub-beams, those that can use the same cage

- **a.** Same size
- **b.** Same covers
For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement is calculated for the range of bars specified in the Design Brief. The program's limits design to 8 bars in any one layer and uses a maximum of 2 layers in each face.

The program then goes along the beam and checks each section to see how many bars from the critical sections can be removed. The bars are only removed at the section if they are not required for compression reinforcement or would result in failure in a crack check.

Design for Shear

The shear reinforcement is designed to resist the major axis shear force envelope, $F_z$, acting through the beam. The minor axis shear and torsional forces are not considered.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $v$, and concrete shear capacity, $v_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.

Design for Torsion

The method for designing a beam with a rectangular section to resist torsion is as follows. It consists of calculations to determine an additional area of longitudinal and link reinforcement required to resist the torsional shear forces.

1. Determine $A_s$ and $A_{sv}$ to resist the bending moments and shear forces by the usual procedures.
2. Calculate the torsional shear stress (for rectangular) clause 2.4.4.1

\[ v_t = \frac{2T}{h_{\min} \left( h_{\max} - h_{\min} / 3 \right)} \]

where

\[ T = \text{torsional moment due to the ultimate loads} \]
\[ h_{\min} = \text{the smaller dimension of the beam section} \]
\[ h_{\max} = \text{the larger dimension of the beam section} \]

3. If \( v_t > v_{t\text{min}} \) in table 2.3, then torsional reinforcement is required. Refer table 2.4 for the reinforcement requirements with a combination of torsion and shear stress \( v \).

- but not more than 0.4 N/mm\(^2\)
- but not more than 5 N/mm\(^2\)

**Table 81: Table 2.4 Reinforcement for shear and torsion**

<table>
<thead>
<tr>
<th>( vt \leq v_{t\text{min}} )</th>
<th>( vt &gt; v_{t\text{min}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( v \leq v_c + 0.4 )</td>
<td>Minimum shear reinforcement; no torsion reinforcement</td>
</tr>
<tr>
<td>( v &gt; v_c + 0.4 )</td>
<td>Designed shear reinforcement; no torsion reinforcement</td>
</tr>
</tbody>
</table>

4. Calculate the additional shear reinforcement required from torsion (as per clause 2.4.7)

\[ S_v < 200 \text{ mm or } x_1 \]

Where \( x_1 \) is smaller center-to-center dimension of a link.

\( y_1 \) is larger center-to-center dimension of a link.

5. Calculate the additional area of longitudinal steel. (as per clause 2.4.7)

For beams formed from Tee and L shaped profiles, each cross section is divided into rectangles so that the stiffness is maximized as defined in clause 2.4.4.2. The rectangle with the largest \( h_{\text{min}}^3 \cdot h_{\text{max}} \) is then checked to ensure that the torsional shear stress does not exceed the limits and the maximum link spacing is based on this rectangle.

**Anchorages and Bond Lengths**

Anchorages and bond lengths are calculated in accordance with the requirements of clause 3.12.8. They can be displayed graphically on the Main Reinforcement diagram and are used for the schedule table and export to Multi-RC.

**Related Links**

- D. BS 8110 Beam Design Brief dialog (on page 1255)
- D. BS 8110 Column Design Brief dialog (on page 1257)
- D. BS8110 Slab Design Brief dialog (on page 1260)

D. BS8110 Column Design Principles

The BS8110 Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.
Design
D. Concrete Design

Design Principles

Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then rechecked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the 'Guiding Load Case', and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The main reinforcing bars may be modified to allow for other bars passing through the section and then rechecked to ensure that the design requirements are satisfied.

Code Clauses Implemented

The following lists the code clauses used from BS 8110 Part 1:1997.

3.8 Column

3.8.1 General. Program Conforms
3.8.1.1 Symbols. Program Conforms
3.8.1.2 Size of columns. N/A
3.8.1.3 Short and slender columns. Program Conforms
3.8.1.4 Plain concrete columns. N/A
3.8.1.5 Braced and unbraced columns. Program Conforms

3.8.1.6 Effective height of a column

3.8.1.6.1 General. Program Conforms with Beta as a user input
3.8.1.6.2 End conditions. Program Conforms with Beta as a user input
3.8.1.7 Slenderness limits for columns. Program conforms
3.8.1.8 Slenderness of unbraced columns. Program conforms

3.8.2 Moments and forces in columns

3.8.2.1 Columns in monolithic frames designed to resist lateral forces. User decision
3.8.2.2 Additional forces induced by deflection at ULS. Program Conforms for column being designed, not for members connected
3.8.2.3 Columns in column and beam construction or in monolithic braced structural frames. Forces to be designed for are determined by the user's own model.
3.8.2.4 Minimum eccentricity. Program conforms

3.8.3 Deflection induced moments in solid slender columns

3.8.3.1 Design
3.8.3.2 Design moments in braced columns bent about a single axis. Program Conforms
3.8.3.3 Slender columns bend about a single axis (major or minor). Program Conforms
3.8.3.4 Columns where le/h exceeds 29, bent about their major axis. Program Conforms
3.8.3.5 Columns bent about their major axis. Program Conforms
3.8.3.6 Slender columns bent about both both axes. Program Conforms
3.8.3.7 Unbraced structures. Program Conforms
3.8.3.8 Deflection of unbraced column. N/A
3.8.3.9 Additional moments on members attached to a slender column. N/A

3.8.4 Design of column section for ULS

3.8.4.1 Analysis of sections. Program Conforms
3.8.4.2 Design charts for symmetrically-reinforced columns. N/A
3.8.4.3 Nominal eccentricity of short columns resisting moments and axial loads. N/A
3.8.4.4 Short braced columns supporting an approximately symmetrical arrangement of beams. N/A
3.8.4.5 Biaxial bending. Program Conforms
3.8.4.6 Shear in columns. Program Conforms
3.8.5 Deflection of columns. N/A
3.8.6 Crack control in columns. The clause states that “if crack check is required, then the member should be checked as a beam”

N/A is to be interpreted as either not application or not implemented.

Clauses that do not appear in the list are deemed not to be considered by the program.

Related Links

- D. BS 8110 Beam Design Brief dialog (on page 1255)
- D. BS 8110 Column Design Brief dialog (on page 1257)
- D. BS8110 Slab Design Brief dialog (on page 1260)

D. BS8110 Slab Design Principles

The BS8110 Slab Design Brief is for any slab modeled with F.E. regardless of boundary restrained conditions.

Slabs are designed for flexure in two orthogonal directions for both hogging and sagging moments. It does not include design for shear or punching shear checks.

Analysis results for plate elements that form a 'Slab' are used to generate design moments with respect to the axis of the slab. These moments are then used to generate required reinforcement contours. An option if available in the slab design brief to use moment theory developed by Wood an Armer ("The Theory of the Strip Method for Design of Slabs” Proceedings, Institution of Civil Engineers, London; Vol. 41, October 1968, pp. 285-313).

The level of accuracy of results depends on the meshing density used to model the slabs. It is recommended that you examine the accuracy of the model at the analysis stage and ensure satisfactory results are obtained prior to the design stage.

Design for Flexure

Longitudinal and transverse reinforcement are calculated for both sagging and hogging moments on the basis of the slab thickness and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a slab is based on an envelope of design forces and thus at each element center, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment in the two orthogonal directions (i.e., Slab/Region local axis).

Reinforcement calculation is based on the BS8110 recommendations for the design of rectangular beams assuming a unit meter width together with the elements thickness.

Slab end restrained conditions are not catered for at the design stage, however to ensure accuracy of design results, they must be incorporated in the analytical model of the structure.
Design
D. Concrete Design

Note: The program does not perform shear checks per BS 8110.

Related Links
- D. BS8110 Slab Design Brief dialog (on page 1260)
- D. Concrete Slab pages (on page 1212)
- D. BS 8110 Beam Design Brief dialog (on page 1255)
- D. BS 8110 Column Design Brief dialog (on page 1257)
- D. BS8110 Slab Design Brief dialog (on page 1260)

**D. American Concrete design per ACI 318 - 1999**

RC Designer has two ACI 318 - 1999 design briefs, the design principles utilized by these are defined in:

<table>
<thead>
<tr>
<th>Code Ref</th>
<th>Description</th>
<th>Revision</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACI 318-99</td>
<td>Building Code Requirements for Structural Concrete (ACI 318-99) and Commentary (ACI 318R-99)</td>
<td>Second printing, February 1996</td>
</tr>
</tbody>
</table>

D. ACI 318 - 99 Beam Design Principles

The ACI 318-99 Beam Design Brief is for single or multi span prismatic, rectangular solid or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to D. Suitable Member Properties (on page 1006) for more details.

Beams are designed for flexure, shear and torsion. Each member is divided into equally spaced sections along its length and the design is performed at each of these locations. The user can specify the number of segments to be considered (between 4 and 25) for each span. Sections are also taken at face of supports and at locations of the maximum positive and negative bending moments.

**Design for Flexure**

The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of the design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment values from the analysis.

The beam is then divided into sub-beams, those that can use the same reinforcement cage and having

a. Same overall beam size
b. Same cover requirements

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement pattern is calculated for the range of bars specified in the Design Brief. The program does not have a limit on the number of bars in any one layer as long as the spacing requirements specified in the code are satisfied. The program can handle a maximum of 4 layers of reinforcement, two each for the top and bottom layers.
The program then performs a check at each of the defined sections to determine the number of bars, if any, that can be curtailed. The reinforcement bars will not be curtailed at these sections in the following cases:

a. If the bars are required for compression or
b. If curtailling these bars would result in a failure of crack-width checks or
c. If curtailling these bars would result in a failure of the minimum reinforcement checks.

Design for Shear and Torsion

Stirrups are designed to resist the major axis shear force envelope, Fz, and optionally the torsional moments induced in the beam. The minor axis shear forces and torsional moments are not considered in the design.

The bar size for shear stirrups and the minimum number of shear legs to be provided are specified in the Design Brief. Therefore the required spacing for minimum stirrups can be defined. The program then checks each section to determine the shear stress, \( v \), and concrete shear capacity, \( v_c \). From this, the section is classified as either ‘minimum stirrup’ or a ‘high shear’ section. Adjacent sections of the same type are grouped into zones. For non minimum stirrup zones, the shear stirrups are designed for the maximum shear force within that zone.

If torsion is to be considered, the program checks whether the threshold torsion has been exceeded at each section and if so determines the corresponding reinforcement required and then adds this to the requirements for shear.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, \( v \), and concrete shear capacity, \( v_c \). From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.
Anchorage and Bond Lengths
Anchorage and bond lengths are calculated in accordance with the requirements of chapter 12. They can be displayed graphically on the main reinforcement diagram and are used for the schedule table.

Code Clauses Implemented
The following lists the code clauses used from ACI 318-99.

Chapter 3 - Materials
3.3.2.(a)
3.3.2.(c)

Chapter 7 - Details of Reinforcement
7.1.1
7.1.2
7.1.3.(a)
7.1.3.(b)
7.2.1
7.2.2
7.6.1
7.6.2
7.6.6.1
7.6.6.2
7.6.6.3
7.6.6.4
7.6.6.5
7.10.5.1
7.10.5.3
7.11.1
7.11.2

Chapter 8 - Analysis and Design - General Considerations
8.5.1
8.5.2
8.10.2

Figure 135: Minimum shear links required, V, for shear forces between these values
8.10.3
8.10.4

Chapter 9 - Strength and Serviceability Requirements
9.1.1
9.3.1
9.3.2.1
9.3.2.2
9.3.2.3
9.3.3
9.4
9.5.2.1

Chapter 10 - Flexure and Axial Loads
10.2.2 but not deep flexural members
10.2.3
10.2.4
10.2.5
10.2.6 parabolic stress block assumed except for T and L-sections
10.2.7 for T and L-sections only
10.3.1
10.3.2
10.3.3
10.3.4
10.4.1
10.5.1
10.5.2
10.5.3
10.6.3
10.6.4
10.6.7
10.9.1
10.9.2
10.9.3
10.10.2
10.11.1
10.11.2 - r always computed
10.11.4
10.11.5
10.11.6
10.12.1 - k controlled by brief
10.12.2
10.12.3
10.12.3.1
10.12.3.2
Design

D. Concrete Design

Chapter 11 Shear and Torsion

11.1.1
11.1.2
11.1.2.1
11.1.3.1
11.2.1.1
11.2.1.2
11.3.1.1
11.3.1.2
11.3.2.3
11.5.2
11.5.3
11.5.4.1
11.5.4.3
11.5.5.1 - minimum shear rft always provided
11.5.5.2
11.5.6.1
11.5.6.2
11.5.6.8
11.6.1
11.6.2.1
11.6.2.4
11.6.3.1
11.6.3.4
11.6.3.5
11.6.3.6
11.6.3.7
11.6.3.8
11.6.3.9
11.6.4.3
11.6.5.1
11.6.5.2
11.6.5.3
11.6.6.1
11.6.6.2
11.6.6.3

Chapter 12 - Development and Splices for Reinforcement.

12.1.1
Notes:
The following settings should be checked in the design brief:

- beta d factor.
- whether or not there is transverse loading.
- stability index Q (must be > 0.05 and ≤ 0.6).
- load case number that causes appreciable sidesway. If columns require different loadcases for sidesway, then they should be designed in separate design groups.
- beta factor for biaxial bending (default 0.65).
• **D. ACI 318-99 Column Design Brief dialog** (on page 1221)

D. ACI 318 - 99 Column Design Principles
The ACI Column Design Brief is for prismatic, rectangular solid or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered, so only single height columns are recommended.

Design Principles
Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated in the Analysis mode and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the 'Guiding Load Case', and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations is also for that combination.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

Code Clauses Implemented
The following lists the code clauses used from ACI 318-99.

**Chapter 3 - Materials**

2.1
3.3.2.(a)
3.3.2.(c)

**Chapter 7 - Details of Reinforcement**

7.1.1
7.1.2
7.1.3.(a)
7.1.3.(b)
7.2.1
7.2.2
7.6.1
7.6.2
7.6.3
7.6.6.1
7.6.6.2
7.6.6.3
7.6.6.4
7.6.6.5
7.10.4.1 - 9
7.10.5.1 - 5
7.10.5.1
7.10.5.3
7.11.1
7.11.2

Chapter 8 - Analysis and Design - General Considerations

8.5.1
8.5.2
8.10.2
8.10.3
8.10.4

Chapter 9 - Strength and Serviceability Requirements

9.1.1
9.3.1
9.3.2.1
9.3.2.2
9.3.2.3
9.3.3
9.4
9.5.2.1

Chapter 10 - Flexure and Axial Loads

10.2.2 but not deep flexural members
10.2.3
10.2.4
10.2.5
10.2.6 parabolic stress block assumed except for T and L-sections
10.2.7 for T and L-sections only
10.3.1
10.3.2
10.3.3
10.3.4
10.3.5
10.3.6
10.4.1
10.5.1
10.5.2
10.5.3
10.6.3
10.6.4
10.6.7
10.8
10.9.1
10.9.2
10.9.3
10.10.2
10.11.1
Chapter 11 Shear and Torsion

11.1.1
11.1.2
11.1.2.1
11.1.3.1
11.2.1.1
11.2.1.2
11.3.1.1
11.3.1.2
11.3.2.2
11.3.2.3
11.5.2
11.5.3
11.5.4.1
11.5.4.3
11.5.5.1 minimum shear rft always provided
11.5.5.2
11.5.6.1
11.5.6.2
11.5.6.8
11.6.1
11.6.2.1
11.6.2.4
11.6.3.1
11.6.3.4
11.6.3.5
11.6.3.6
11.6.3.7
11.6.3.8
11.6.3.9
11.6.4.3
Design
D. Concrete Design

Chapter 12 - Development and Splices for Reinforcement.

12.1.1
12.1.2
12.2.2
12.2.3
12.2.4
12.2.5
12.3.1
12.3.2
12.3.3
12.3.3.1
12.3.3.2
12.4.1
12.4.2
12.5.1
12.5.2
12.5.3
12.5.3.1
12.5.3.2
12.5.3.3
12.5.3.4
12.5.3.5
12.5.3.6
12.5.4
12.5.5
12.10.2
12.10.3
12.10.4
12.10.5
12.10.5.1
12.10.5.2
12.10.5.3
12.11.1
12.11.2
12.11.3
12.12.2
12.12.3
Notes:
The following settings should be checked in the design brief:

- β d factor.
- whether or not there is transverse loading.
- stability index Q (must be > 0.05 and ≤ 0.6).
- load case number that causes appreciable sidesway. If columns require different loadcases for sidesway, then they should be designed in separate design groups.
- β factor for biaxial bending (default 0.65).

Related Links
- D. ACI 318-99 Column Design Brief dialog (on page 1221)

D. American Concrete design per ACI 318 - 2005
RC Designer has three ACI 318 - 2005 design briefs, the design principles utilized by these are defined in:

**Code Ref**  ACI 318-2005

**Description**  Building Code Requirements for Structural Concrete (ACI 318-05) and Commentary (ACI 318R-05)

**Revision**  First printing, December 2004

**Note:** [Metric](on page 1105) is available for use with ACI 318M-05 in STAAD.Pro V8i (SELECTseries 2) 20.07.07 and higher.

D. ACI 318-05 Beam Design Principles
The ACI 318-05 Beam Design Brief is for single or multi-span prismatic, rectangular solid, or tee shaped members. The member sections must be defined as PRISMATIC sections in the STAAD.Pro data file.

Refer to [D. Suitable Member Properties](on page 1006) for more details.

Beams are designed for flexure, shear and torsion. Each member is divided into equally spaced sections along its length and the design is performed at each of these locations. The user can specify the number of segments to be considered (between 4 and 25) for each span. Sections are also taken at face of supports and at locations of the maximum positive and negative bending moments.

**Design for Flexure**
The main (longitudinal) reinforcement is calculated for both sagging and hogging moments on the basis of the section profile and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a beam is based on an envelope of the design forces and thus at each of the defined sections, the program determines the required area of steel for both the maximum hogging moment and maximum sagging moment values from the analysis.

The beam is then divided into sub-beams, those that can use the same reinforcement cage and having

- a. Same overall beam size
- b. Same cover requirements

For each sub-beam, the sections that have the largest sagging and hogging moments are identified and the most efficient reinforcement pattern is calculated for the range of bars specified in the Design Brief. The program
The program then performs a check at each of the defined sections to determine the number of bars, if any, that can be curtailed. The reinforcement bars will not be curtailed at these sections in the following cases:

- If the bars are required for compression
- If curtailing these bars would result in a failure of crack-width checks
- If curtailing these bars would result in a failure of the minimum reinforcement checks.

**Design for Shear and Torsion**

Stirrups are designed to resist the major axis shear force envelope, $F_z$, and optionally the torsional moments induced in the beam. The minor axis shear forces are not considered in the design.

The bar size for shear stirrups and the minimum number of shear legs to be provided are specified in the Design Brief. Therefore the required spacing for minimum stirrups can be defined. The program then checks each section to determine the shear force $V_n$ and concrete shear capacity $V_c$. From this, the section is classified as either 'minimum stirrup' or a 'high shear' section. Adjacent sections of the same type are grouped into zones. For non minimum stirrup zones, the shear stirrups are designed for the maximum shear force within that zone.

If torsion is to be considered, the program checks whether the threshold torsion has been exceeded at each section and if so determines the corresponding reinforcement required and then adds this to the requirements for shear.

The number of shear legs and the shear link size is specified in the Design Brief. Therefore the required spacing for minimum links can be defined. The program then checks each section to determine the shear stress, $v$, and concrete shear capacity, $v_c$. From this, the section is classified as either minimum link or a high shear section. Adjacent sections of the same type are grouped into zones. For non minimum link zones, the shear links are designed for the maximum shear force within that zone.

If necessary, additional legs may be added to the shear links in order to restrain tension or compression reinforcement.
Development Lengths

Development lengths are calculated in accordance with the requirements of chapter 12. They can be displayed graphically on the main reinforcement diagram and are used for the schedule table and export to rebar detailing programs.

Code Clauses Implemented

The following lists the code clauses used from ACI 318-05.

Chapter 3 - Materials

3.3.2.(a)
3.3.2.(c)

Chapter 7 - Details of Reinforcement

7.1.1
7.1.2
7.1.3.(a)
7.1.3.(b)
7.1.3.(c)
7.2.1
7.2.2
7.6.1
7.6.2
7.11.1
7.11.2 – If selected in the design brief

Chapter 8 - Analysis and Design - General Considerations

8.5.1
8.5.2
8.10.2
8.10.3
8.10.4

Chapter 9 - Strength and Serviceability Requirements

9.1.1
9.3.1
9.3.2.1
9.3.2.2 (b)
9.3.2.3
9.3.3
9.4
9.5.2.1

Chapter 10 - Flexure and Axial Loads

10.2.2 but not deep beams
10.2.3
10.2.4
10.2.5
10.2.6 parabolic stress block assumed except for T and L-sections
10.2.7 for T and L-sections only
10.3.1
10.3.2
10.3.3
10.3.4
10.4.1
10.5.1
10.5.2
10.6.3
10.6.4
10.6.7

Chapter 11 Shear and Torsion

11.1.1
11.1.2
11.1.2.1
11.1.3.1 – If selected in brief
11.2.1.1
11.2.1.2
11.3.1.1
11.3.1.2
11.3.2.3
11.5.2
11.5.3
11.5.5.1
11.5.5.3
11.5.6.1 - minimum shear reinforcement always provided
11.5.6.3
11.5.7.1
11.5.7.2
11.5.7.9
Design
D. Concrete Design

11.6.1
11.6.2.1
11.6.2.4
11.6.3.1
11.6.3.4
11.6.3.5
11.6.3.6
11.6.3.7
11.6.3.8
11.6.4.3
11.6.5.1
11.6.5.2
11.6.5.3
11.6.6.1
11.6.6.2

Chapter 12 - Development and Splices for Reinforcement.

12.1.1
12.1.2
12.2.1
12.2.2
12.2.3
12.2.4
12.2.5
12.3.1
12.3.2
12.3.3 (a)
12.3.3.1
12.4.1
12.4.2
12.5.1
12.5.2
12.5.3
12.5.3. (a)
12.5.3. (b)
12.5.3. (c)
12.5.3. (d)
12.5.4
12.5.5
12.10.2
12.10.3
12.10.4
12.10.5
12.10.5.1
12.10.5.2
12.10.5.3
D. ACI 318-05 Column Design Principles
The ACI 318-05 Column Design Brief is for prismatic, solid rectangular or circular shaped members. Members may be built up from multiple elements, but only the member end moments are considered. Hence only single height columns are recommended.

Design Principles
Columns are designed for axial force and biaxial end moments, as well as biaxial shear. Torsional moments are also included if specified in the design brief. Reinforcement is provided by the program based on the forces generated from the analysis model and the code clauses outlined below.

The main reinforcing bars may be modified to allow for other bars passing through the section and then re-checked to ensure that the design requirements are satisfied.

All selected load combinations are used to calculate the required reinforcement. The combination which produces the maximum required reinforcement area is called the 'Guiding Load Case', and the bar arrangement is chosen based on that combination. Detailed printout of the design calculations provided is for that combination.

Code Clauses Implemented
The following lists the code clauses used from ACI 318-05.

Chapter 3 - Materials
2.1
3.3.2.(a)
3.3.2.(c)

Chapter 7 - Details of Reinforcement
7.1.3 (a)
7.1.3.(b)
7.2.1
7.2.2
7.6.1
7.6.3
7.10.4.2
7.10.4.3
7.10.4.4
7.10.5.1 - 3
7.10.5.1
7.10.5.3
7.11.2
Chapter 8 - Analysis and Design - General Considerations

8.5.1
8.5.2
8.10.2
8.10.3
8.10.4

Chapter 9 - Strength and Serviceability Requirements

9.1.1
9.3.1
9.3.2.1
9.3.2.2
9.3.2.3
9.3.3
9.4

Chapter 10 - Flexure and Axial Loads

10.2.2 but not deep flexural members
10.2.3
10.2.4
10.2.5
10.2.6
10.2.7
10.3.1
10.3.2
10.3.3
10.3.4
10.3.6.1
10.3.6.2
10.3.7
10.5.1
10.6.3
10.6.4
10.9.1
10.9.2
10.10.2
10.11.1
10.11.2
10.11.4
10.11.5
10.11.6
10.12.1 k controlled by brief
10.12.2
10.12.3
10.12.3.1
Chapter 11 Shear and Torsion

11.1.1
11.1.2
11.1.2.1
11.2.1.1
11.2.1.2
11.3.1.1
11.3.1.2
11.3.2.2
11.3.2.3
11.3.3
11.5.1(a)
11.5.1 (c)
11.5.2
11.5.5.1
11.5.6.1 minimum shear rft always provided
11.5.6.3
11.5.7.1
11.5.7.2
11.5.7.9
11.6.1
11.6.2.1
11.6.3.1
11.6.3.4
11.6.3.5
11.6.3.6
11.6.3.7
11.6.3.8
11.6.4.1
11.6.4.2
11.6.5.1
11.6.5.2
11.6.5.3
11.6.6.1
11.6.6.2
**Notes:**

The following settings should be checked in the design brief:

- beta_d factor.
- whether or not there is transverse loading.
- stability index \( Q \) (must be \( > 0.05 \) and \( \leq 0.6 \)).
- load case number that causes appreciable sidesway. If columns require different loadcases for sidesway, then they should be designed in separate design groups.
- beta factor for biaxial bending (default 0.65).

**Related Links**

- [D. ACI 318-05 and 318M-05 Column Design Brief dialog](on page 1230)
- [D. ACI 318-05 and 318M-05 Slab Design Brief dialog](on page 1235)
- [D. ACI 318-05 and 318M-05 Beam Brief dialog](on page 1224)

D. ACI 3 18-05 Slab Design Principles

The ACI 3 18-05 Slab Design Brief is for any slab modeled with finite elements regardless of boundary restraint conditions.

Slabs are designed for flexure in two orthogonal directions for both hogging and sagging moments. Designs for shear or punching shear checks are not included.

Analysis results for plate elements that form a 'Slab' are used to generate design moments with respect to the axes of the slab. These moments are then used to generate required reinforcement contours. Note that no account is taken of the requirements for embedment or development of reinforcement.

**Tip:** The level of accuracy of results depends on the meshing density used to model the slabs. It is recommended that users examine the accuracy of the model at the analysis stage and ensure satisfactory results are obtained prior to the design stage.

**Design for Flexure**

Longitudinal and transverse reinforcement are calculated for both sagging and hogging moments on the basis of the slab thickness and parameters defined in the Design Brief. Compression reinforcement is provided where required.

The design of a slab is based on an envelope of design forces. Hence the program determines the required area of steel at the center of each element, for both the maximum hogging and sagging moments in the two orthogonal directions. (i.e., along the Slab/Region local axes)

Reinforcement calculation is based on the ACI 318-05 recommendations for the design of one-way slab systems assuming a unit width together with the elements thickness.

Slab end restraint conditions are not catered for at the design stage, however to ensure accuracy of design results, they must be incorporated in the analytical model of the structure.

As has been stated earlier, in this release of the RC designer, shear checks have not been catered for.

**Code Clauses Implemented**

The list of code clauses used from ACI 318-05 is as follows:

Chapter 3 - Materials

3.3.2 (c)
Chapter 7 - Details of Reinforcement

7.6.1
7.6.5
7.12.2.1

Chapter 9 - Strength and Serviceability Requirements

9.1.1
9.3.1
9.3.2.1 – Sections are assumed to be tension controlled

Chapter 10 - Flexure and Axial Loads

10.2.2 but not deep flexural members
10.2.3
10.2.4
10.2.7
10.5.4 – but with spacing not greater than twice the slab thickness (assumes two-way slab as 13.2.2)
10.6.4

Chapter 13 – Two Way Slab Systems

13.2.2

Related Links

- D. ACI 318-05 and 318M-05 Column Design Brief dialog (on page 1230)
- D. ACI 318-05 and 318M-05 Slab Design Brief dialog (on page 1235)
- D. ACI 318-05 and 318M-05 Beam Brief dialog (on page 1224)

D. American Concrete design per ACI 318M - 2005

RC Designer has three ACI 318M - 2005 design briefs, the design principles utilized by these are defined in:

Note: This feature is available in STAAD.Pro V8i (SELECTseries 2) release 20.07.07 and higher.

Code Ref ACI 318M-2005
Description Metric Building Code Requirements for Structural Concrete (ACI 318M-05) and Commentary (ACI 318R-05)
Revision First printing, January 2005

See D. American Concrete design per ACI 318 - 2005 (on page 1096) for information on the implementation of ACI 318-05.

D. Earthquake Collapse Check

This checks at each column / beam interface, the program checks that the capacity of the column exceeds the total capacity of all beams that connect to it.

The earthquake check only uses the results from Design Groups that have Design Briefs from the selected Design Code.
D. Application Window Layout

This section documents all aspects of the Concrete Design program interface.

The application window contains the following special sections by default:

A. **Menus** (on page 1106)
B. **Standard toolbar** (on page 1193)
C. **Page/Mode Automatic toolbar** (on page 1195)
D. **Page Control** (on page 1202)
E. **View Window** (on page 1014)
F. **Status Bar** (on page 1217)

---

D. Menus

These are the commands that are available from the menus.

**Note:** The menus and items change depending on the mode and the type of current window (e.g., Graphic or Table).
D. File menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Opens the link file and ignores the data in the persistent file.</td>
<td>&lt;Ctrl+N&gt;</td>
</tr>
<tr>
<td>Re-open</td>
<td>Opens the link file and includes the data in the persistent file. Effectively restarts the Concrete Design, removing any data that has been added since the current RC Designer session started.</td>
<td>&lt;Ctrl+O&gt;</td>
</tr>
<tr>
<td>Save</td>
<td>Saves the data that has been generated in the Concrete Design in a persistent file &lt;filename&gt;.Rei_Concrete. &lt;Tip: To save a structure quickly, click the Save button on the Standard toolbar.&gt;</td>
<td>&lt;Ctrl+S&gt;</td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
<td>Shortcut</td>
</tr>
<tr>
<td>----------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Job Information</td>
<td>Toggles the display of the <a href="#">Job Information dialog</a> (on page 1109) for the active structure, which is used to provide general information and comments about the structure.</td>
<td></td>
</tr>
<tr>
<td>Report Setup</td>
<td>Opens the <a href="#">Report Setup dialog</a> (on page 1110), which is used to control the contents and look of your reports. The Report Setup command is sensitive to the current mode and to the current design code if in Design mode. That is, the menu item will display text based on the current design brief and elements selected.</td>
<td></td>
</tr>
<tr>
<td>Printer Setup</td>
<td>Displays the <a href="#">Print Setup dialog</a> (on page 1114), which provides a list of installed printers, sets the default printer, and provides access to other printing options for the printer selected.</td>
<td></td>
</tr>
<tr>
<td>Print</td>
<td>Opens the <a href="#">Print dialog</a> (on page 1114), which controls how the Concrete Design report is printed.</td>
<td>&lt;Ctrl+P&gt;</td>
</tr>
<tr>
<td>Print Preview</td>
<td>Opens the <a href="#">Print Preview window</a> (on page 1115), which displays how the Concrete Design report appears when printed.</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** Before using this command, install and select a printer. For more information refer to the Windows documentation.

The appearance and content of the Concrete Design report can be configured using **File > Report Setup**. The Printer settings are modified by selecting **File > Print Setup**.
The menu is available in the modes:

- Design Layer
- Member Design
- Slab Design
- Earthquake

**D. Job Information** dialog

Used to provide general information and comments about the structure. When first opened, the dialog is populated with the data that is in the STAAD.Pro input file.

Some of the Job Information is printed in the header on every page of the reports. The rest may be printed in the special Job Information report item.

Opens when:

- **File > Job Information** is toggled on (a check is shown), or
- When **Des. Layer | Job Info** (on page 1202) page is selected.
Job
The job title (up to 100 characters) is printed in the report header.

Client
The client name (up to 100 characters) is printed in the report header.

Job No.
The job number (up to 100 characters) is printed in the report header.

Part
The part reference (up to 100 characters) is printed in the report header.

Ref
The reference (up to 100 characters) is printed in the report header.

Quality Assurance
Six edit boxes are available for recording information required by quality assurance procedures. There are spaces (10 characters) for the names or initials of the Engineer, the Checking Engineer and the Approving Engineer, together with dates.

Comment
Up to 1000 characters of comments can be typed into this edit box. The comments appear on the special Job Information report item.

D. (code) Report Setup dialog
Used to control the contents and look of your reports. The Report Setup command is sensitive to the current mode and to the current design code if in Design mode. That is, the menu item will display text

Opens when:

• File > (code) Report Setup is selected, or
the Report Setup tool is selected

Items tab

Groups Select the previously defined Design Group for which you want to generate a report.

Available list The Available list shows the tables and pictures that can be included in the report.

Tip: More than one item can be selected at once by holding down the CTRL key while selecting. A range of items can be selected by click and dragging up or down the list or clicking on the first item and clicking on the last whilst holding down the SHIFT key.

Selected list Results, tables, and pictures in this list will be included in the report, in the order in which they appear.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

The up and down arrows beside the Selected list can be used to re-order items in the report.
Detailed Results tab

**Available**  The Available list shows the detailed items that *can* be included in the report if the ‘Design Detail’ item is included in the Items list (see above).

**Selected**  The Selected list shows the detailed items that *will* be included in the report if the ‘Design Detail’ item is included in the Items list (see above).

Members tab

**Available**  The Available list shows the members that *can* be included in the report.

**Selected**  The Selected list shows the members that *will* be included in the report.

Member Loadcases tab

Not available for all codes.

**Available list**  Contains all primary load cases and load combinations in the STAAD.Pro input file.

**Selected list**  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the <strong>Selected</strong> list.</td>
</tr>
</tbody>
</table>

**Note:** By default, all load cases or combinations are selected.

Design Briefs tab

**Available**  The Available list shows the detailed items that *can* be included in the report if the ‘Other Brief’s Details’ item is included in the Items list (see above).

**Selected**  The Selected list shows the detailed items that *will* be included in the report if the ‘Other Brief’s Details’ item is included in the Items list (see above).

Picture Album tab

Shows the pictures that have been captured for inclusion in the report. It also allows the captions for these pictures to be edited.

**Name**  Select the picture to view using the drop-down list.

**Caption**  Edit the caption for the picture in this edit box. The caption appears printed below the picture in the report.

**Delete**  Removes the picture from the picture album.

**Full Page**  Select this option to set the current picture to take a full page in the printout if it is an included item.

Options tab
Sets options for how the printed pages of the report appear.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Header</td>
<td>Select this option to include a header on the report. Note that the header contains the Sheet Numbering, company Name and Logo, and much of the Job Info.</td>
</tr>
<tr>
<td>Page Outline</td>
<td>Select this option to add an outline to each page of the report.</td>
</tr>
<tr>
<td>Footer</td>
<td>Select this option to include a footer on the report. Note that the print run information, that is page numbers, time and date, is included on the footer.</td>
</tr>
<tr>
<td>Prefix</td>
<td>Sheet numbers are printed at the top right of each page. Edit the prefix for sheet numbers here. The prefix is printed one space in front of the sheet number.</td>
</tr>
<tr>
<td>No. Pages From</td>
<td>Give a number from which to start the sheet numbering.</td>
</tr>
<tr>
<td>Suffix</td>
<td>Edit the suffix for the sheet numbers at the top right of the page. Leave this text box empty if a suffix is not needed.</td>
</tr>
<tr>
<td>Reverse Page Order</td>
<td>Some printers do not turn over the pages that they have printed. This results in a printed report coming out back to front. If your printer does this, then select the Reverse Page Order option and QSE Space will print the last page first!</td>
</tr>
<tr>
<td>Grid</td>
<td>Select this option to draw the grid lines in the data tables of the report. Leaving the grid lines off increases printing speed on some printers.</td>
</tr>
<tr>
<td>Start each table on a new page</td>
<td>Select this option to start each new data table on a new report page. This option is useful for reports on very large structures, but it wastes paper on small structures where the tables are short.</td>
</tr>
<tr>
<td>Table Heading</td>
<td>Click on the Font... button to show the Font dialog for the table headings in the report. Choose a new font using the dialog and click the OK button to return to the Report Setup dialog.</td>
</tr>
<tr>
<td>Table Text</td>
<td>Click on the Font... button to show the Font dialog for the table text in the report. Choose a new font using the dialog and click the OK button to return to the Report Setup dialog.</td>
</tr>
</tbody>
</table>

Name and Logo tab

Configures the company name and logo area of the report page. This is situated in the box between the Bentley Systems logo (top left) and the Job No. box.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Preview</td>
<td>Shows a preview of how the company name and logo appear. The company name is edited directly by typing in the preview.</td>
</tr>
<tr>
<td>File …</td>
<td>Click on the File button to insert a Windows Metafile graphic file into the company logo area.</td>
</tr>
<tr>
<td>Paste</td>
<td>Click to paste a bitmap or Windows Metafile image from the clipboard into the company logo area of the report header. If the button is inactive, the clipboard does not contain graphical data.</td>
</tr>
<tr>
<td>Position</td>
<td>Adjust the position of the graphic using these options. If the graphic is placed in the center, then the company name does not appear.</td>
</tr>
<tr>
<td>Font…</td>
<td>Opens a Font dialog, which is used to change the font and size of the text used for the company name. The text itself is edited directly in the Preview area.</td>
</tr>
<tr>
<td>Alignment</td>
<td>Adjust the alignment of the company name using these options. The company name is aligned within the space left after the company logo (if any) is drawn. Note that if the logo is placed in the center then the company name does not appear (although it is not deleted).</td>
</tr>
</tbody>
</table>
Load/Save tab

Used to save a report setup which can be recalled at a later date. The arrangement of Items, Members and Design Briefs with the sort options are saved.

When a picture is deleted, it is removed from all reports.

A Saved Report Setup can be deleted.

D. Print Setup dialog

Provides a list of installed printers, sets the default printer, and provides access to other printing options for the printer selected.

Opens when File > Print Setup… (on page 1107) is selected.

![Print Setup dialog]

**Default Printer**  Displays the name of the default printer and shows the printer’s connection.

**Printers**  Selects the printer to be used. Only printers that have been installed are listed here. For more information refer to the Windows documentation.

**Properties...**  Changes the printer settings from the printer selected in the Printers list. The options vary according to the printer driver installed.

**Network...**  Connects the computer to a printer across a network. For more information refer to the Windows documentation or press Help in the Network dialog.

**Orientation**  Select the orientation which you wished to be used for the printed output.

**OK**  Applies the Print Setup options and closes the dialog.

**Cancel**  Closes the dialog without making any changes.

D. Print dialog

Controls how the Concrete Design report is printed.

Opens when File > Print is selected.
**Printer**  Displays the name of the active printer and printer connection.

**Properties...**  Changes the printer settings from the printer selected in the Printers list. The options vary according to the printer driver installed.

**Print Range**  Select how much of the report is to be printed.

**Setup**  Displays the Print Setup dialog. This is equivalent to choosing the Printer Setup command from the File menu.

**OK**  Prints the report to the selected printer and closes the dialog.

**Cancel**  Closes the dialog without printing.

**D. Print Preview window**
Displays how the Concrete Design report appears when printed.

The arrangement and contents of a report are modified in the Report Setup dialog (on page 1110).

Open when **File > Print Preview** is selected.

When entering the print preview, Concrete Design repaginates the report and retrieves all the current data it needs, so the preview is the same as would be printed.
Design

D. Concrete Design

Design Group: - Brief Detail: BS8110 Slab Brief

Group Data

<table>
<thead>
<tr>
<th>Top Cover</th>
<th>Bottom Cover</th>
<th>Aggregate Size</th>
<th>Concrete Strength Class</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td>30</td>
<td>30</td>
<td>C4</td>
</tr>
<tr>
<td>Envelope</td>
<td>Envelope 1</td>
<td>Wood and Armour</td>
<td>No</td>
</tr>
<tr>
<td>Sampling</td>
<td>0.200 m</td>
<td></td>
<td>Design Type</td>
</tr>
</tbody>
</table>

Main Reinforcement

Main bars: HRB 500

Main Bar Criteria

<table>
<thead>
<tr>
<th>Outer Bar Direction</th>
<th>Top Bar Criteria</th>
<th>Bottom Bar Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
</tbody>
</table>

Bar Size X: 20

Bar Size Y: 20

Slab Information

<table>
<thead>
<tr>
<th>Gross Area</th>
<th>72.000 m²</th>
<th>Number of Plates</th>
<th>32</th>
</tr>
</thead>
<tbody>
<tr>
<td>Net Area</td>
<td>72.000 m²</td>
<td>Number of Holes</td>
<td>0</td>
</tr>
<tr>
<td>Thickness</td>
<td>15.0 cm</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Region Information

<table>
<thead>
<tr>
<th>Region</th>
<th>Name</th>
<th>Orientation (°)</th>
<th>Area (m²)</th>
<th>Thickness (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reg 1</td>
<td>Region 1</td>
<td>90.00</td>
<td>36,000</td>
<td>15.0</td>
</tr>
<tr>
<td>Reg 2</td>
<td>Region 2</td>
<td>0.00</td>
<td>36,000</td>
<td>15.0</td>
</tr>
</tbody>
</table>

Design Summary

<table>
<thead>
<tr>
<th>Region</th>
<th>Name</th>
<th>Direction</th>
<th>Design. Mom. (KN/m)</th>
<th>Min. Steel</th>
<th>Bar Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reg 1</td>
<td>Region 1</td>
<td>Top X</td>
<td>20.620</td>
<td>n</td>
<td>H20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Top Y</td>
<td>11.066</td>
<td>n</td>
<td>H20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Bottom X</td>
<td>12.927</td>
<td>n</td>
<td>H20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Bottom Y</td>
<td>12.927</td>
<td>n</td>
<td>H20</td>
</tr>
<tr>
<td>Reg 5</td>
<td>Region 2</td>
<td>Top X</td>
<td>11.066</td>
<td>n</td>
<td>H20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Top Y</td>
<td>26.686</td>
<td>n</td>
<td>H20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Bottom X</td>
<td>12.927</td>
<td>n</td>
<td>H20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Bottom Y</td>
<td>12.927</td>
<td>n</td>
<td>H20</td>
</tr>
</tbody>
</table>
Design
D. Concrete Design

Print... Opens the Print dialog (on page 1114).

Next / Prev Page Moves the view forward and backward, respectively, through the report preview.

One / Two Page Toggles the display of the report preview between an single page and two page layout.

Zoom In / Out Magnifies or decreases the zoom level of the report preview.

Close Closes the Print Preview window and returns the to RC Designer window.

D. Edit menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo [action]</td>
<td>Reverses the last change made to the active structure. The name of the command depends on the most recent action taken (e.g., Undo Delete). Note: Concrete Design cannot undo some actions. If the most recent action cannot be undone, then the name of the command reverts to Undo and it is disabled. Tip: To quickly reverse the most recent action on the active structure, click on the Undo button on the Standard toolbar.</td>
<td>Ctrl+Z</td>
</tr>
<tr>
<td>Cut</td>
<td>Removes the selected table data and puts it on to the Clipboard. Data cut to the clipboard replaces the previous contents of the clipboard. It remains on the clipboard until new data is cut or copied. This command is only available once a valid selection has been made in the active window. Tip: To quickly cut information to the clipboard, make a selection and then click the Cut button on the Standard toolbar.</td>
<td>Ctrl+X</td>
</tr>
</tbody>
</table>
### Menu item | What it does | Shortcut
---|---|---
**Copy** | Copies the selected table data and puts it on to the Clipboard. Data that is copied to the clipboard replaces the previous contents of the clipboard. It remains on the clipboard until new data is cut or copied. This command is only available once a valid selection has been made in the active window. **Tip:** To quickly copy information to the clipboard, make a selection and then click the Copy button on the Standard toolbar. | Ctrl+C

**Copy Picture** | Copies the picture in the active structure window to the clipboard. No selection is needed for this command to work. The picture replaces the previous contents of the clipboard and can be pasted into any windows application that supports graphics (e.g., Microsoft Office Word Excel). | 

**Paste** | Inserts a copy of the data on the clipboard into the active window. If the data on the clipboard is empty or is unsuitable for pasting into the active window, then the command is disabled. | Ctrl+V
### Menu item

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delete</td>
<td>Deletes the selection in the active window. A selection must be made for this command to be available. If the current window is a structure, a warning message is shown allowing the command to be canceled if required.</td>
<td>Delete</td>
</tr>
<tr>
<td>Tip:</td>
<td>If a mistake is made, use the Undo command on the Edit menu to reverse the delete. This must be done immediately after the mistake or the Undo may not be available.</td>
<td></td>
</tr>
<tr>
<td>Take Picture</td>
<td>Opens the Picture # dialog (on page 1120), which is used to copy the active structure window to an image for including in the report picture album. No selection is needed for this command to work. Photographs are stored for future use in a separate file in the same folder as the structure file with the same filename but with the file extension .REI Saved Picture.</td>
<td></td>
</tr>
<tr>
<td>Go To</td>
<td>Opens the GoTo Row dialog (on page 1120), which is used to scroll the active table window so that a given row is visible.</td>
<td>F5</td>
</tr>
</tbody>
</table>

The menu is available in the modes:

- Design Layer
- Member Design
- Slab Design
- Earthquake

D. Using table rows
When pasting into a table, any selected rows are overwritten. The paste command for a table keeps the data in the same order as it is on the clipboard.

If the data on the clipboard is longer than the selection in the active table, then the paste fills in any blank rows before appending the remaining data to the end of the table.

If the data on the clipboard is narrower than the active table, the data is pasted into the columns on the left of the table, leaving other columns unchanged or displaying default values.
If paste comes across an empty field in the clipboard data, it leaves any existing data alone, or enters a default value.

D. **Goto Row** dialog
Used to specify a row number for easy selection.

Opens when **Edit > Go To** is selected.

![Goto Row dialog](image)

**Row Number** Specify the row number (typically corresponds to beam, slab, element, etc. number) you want to select.

**OK** Selects the specified row number and closes the dialog.

**Cancel** Closes the dialog without selecting any other row.

D. **Picture #** dialog
Used to copy the active structure window to an image for including in the report picture album.

Opens when:
- **Edit > Take Picture** is selected, or
- The **Take Picture** tool is selected.

![Picture dialog](image)

**Id** Specify a title used to identify the photo in the Items list and Picture Album of a Report Setup (on page 1110).

**Caption** (Optional) Specify a caption to be used if the picture is a selected item in the Report Setup.

**OK** Accepts the changes made and closes the dialog.
**Cancel**  Closes the dialog without making any changes.

**D. View menu**

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Zoom</strong></td>
<td>Opens a Zoom Window onto the active structure window.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>When the Zoom command is chosen, the cursor changes to a cross (+).</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Define a zoom area on the structure window. Press down the left mouse button on one corner of this area, hold down the button and move the mouse (drag) to open up the area. The zoom area is outlined with a dotted box as it is dragged. Release the mouse button when the area is the desired size.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>A &lt;Zoom&gt; window is now opened showing the zoom area enlarged. If the Highlight Zoom Area option is checked in the options dialog, a box appears on the structure window, locating the extents of the zoom window.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Only one zoom window per structure window can be open. A zoom window cannot be opened from another zoom window. However, the scale of zoomed area can be increased or decreased by clicking on the ‘+’ or ‘-’ buttons on the bottom right of the zoom window.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
<td>Shortcut</td>
</tr>
<tr>
<td>-------------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Tables</td>
<td>Opens the [Tables dialog](on page 1126), which is used to control the display of the table windows in Concrete Design.</td>
<td></td>
</tr>
<tr>
<td>Structure Diagram</td>
<td>Opens the [Diagrams (Structure) dialog](on page 1127), which is used to control the display of the structure, loads, and other graphical items for viewing and reports.</td>
<td></td>
</tr>
<tr>
<td>Colors</td>
<td>Opens the [Colour Manager dialog](on page 1132), which is used to select graphics colors for various plot elements.</td>
<td></td>
</tr>
<tr>
<td>Design Results</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Orientation</td>
<td>Opens the [Orientation dialog](on page 1133), which is used to adjust the orientation and projection of the picture of the structure in the active structure window.</td>
<td>F4</td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> The Orientation dialog floats on above of the other windows in Concrete Design, allowing changes to be seen in the active window if the Apply Immediately check box is selected.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
<td>Shortcut</td>
</tr>
<tr>
<td>------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Labels</td>
<td>Opens the [Diagrams (Structure) dialog](on page 1127) to the labels tab.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> You can toggle the display of the [Labels toolbar](on page 1201) to have quick access to commonly used label tools.</td>
<td></td>
</tr>
<tr>
<td>Whole Structure</td>
<td>Creates a new view window displaying the whole structure view.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>The program can display multiple whole structure windows of the structure, each showing different information (e.g., different load cases, bending moments, displacements, etc.). They may be created at any time. However, the next time the structure file is opened, only one whole structure window is shown.</td>
<td></td>
</tr>
<tr>
<td>Open View</td>
<td>Opens the [Open View dialog](on page 1134), which is used to open a saved view of the structure.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
<td>Shortcut</td>
</tr>
<tr>
<td>-----------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>New View</td>
<td>Creates a new view window from the selection of elements in the active window. One or more elements must be highlighted in the active window for this menu item to become available. An option dialog opens to confirm if you want to replace the modify the current window or open a new window for the new view.</td>
<td></td>
</tr>
<tr>
<td>Tip: If elements are accidentally omitted from the new view, use Add to View If elements are to be excluded from the new view, select the required elements, then use New View to create a corrected view. Close the view window which shows the incorrect selection.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>View Management</td>
<td>Detach View</td>
<td></td>
</tr>
<tr>
<td>Detach View</td>
<td>Detaches the active view from the active structure's data file. Once a view is detached, it regains a title with <strong>&lt;Untitled&gt;</strong> in it and is lost when closing the view window, the data file or exiting Concrete Design. <strong>Note:</strong> A view window must be active for this menu item to be available.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
<td>Shortcut</td>
</tr>
<tr>
<td>-------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Add To View</td>
<td>Opens the [Add to View dialog](on page 1135), which is used to add elements which are selected in the active window to another view window which has already been created. It does not matter if selected elements already exist in the target view, as they remain selected. <strong>Note:</strong> One or more elements must be highlighted in the active window before the Add to View... menu option becomes available.</td>
<td></td>
</tr>
<tr>
<td>Save View</td>
<td>Opens the [Save View As dialog](on page 1136), which is used to save the view in the active window in the data file for the active structure. The view window can then be closed without loosing the view. Use Save View to save a copy of the view in the active window under a new name. A view window must be active for this command to be available.</td>
<td></td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
<td>Shortcut</td>
</tr>
<tr>
<td>-----------</td>
<td>--------------</td>
<td>----------</td>
</tr>
<tr>
<td>Rename View</td>
<td>Opens the Rename View dialog (on page 1136), which is used to rename the view in the active window.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Before a view can be renamed, it must be saved.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> A saved view must be active for this command to be available.</td>
<td></td>
</tr>
<tr>
<td>Toolbars</td>
<td>Opens the Toolbar Setup dialog (on page 1137), which is used to open and configure toolbars.</td>
<td></td>
</tr>
<tr>
<td>Options</td>
<td>Opens the Options dialog (on page 1138), which is used to configure many of the graphical features of Concrete Design.</td>
<td></td>
</tr>
<tr>
<td>Diagram</td>
<td>Opens the Diagrams dialog (on page 1144), which is used to control the display of slab contour diagrams.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> This menu item is only available from the Concrete Slab mode (on page 1212).</td>
<td></td>
</tr>
</tbody>
</table>

The menu is available in the modes:

- Design Layer
- Member Design
- Slab Design
- Earthquake

D. **Tables** dialog
Used to toggle the display of tables in the RC Designer interface.

Opens when:

- **View > Tables** (on page 1121) is selected, or when
• **Other Tables** is selected from a table right-click menu.

![Table Selection Dialog]

**Tables list** Select the tables you want to display by clicking the check box associated with each.

- In the case of the Load Summary table, the Tables dialog affects only the load summary associated with the active structure window.

**List** Select to display Design Input and/or Design Results by clicking the check box associated with each. These options can be used to limit the list of tables to help find items more easily.

**OK** Closes the dialog and the selected tables are displayed.

**Cancel** Closes the dialog without toggling the display of any tables.

**D. Diagrams (Structure) dialog**

Used to control the display of the structure, loads, and other graphical items for viewing and reports.

Opens when:

- **View > Structure Diagrams** (on page 1121) is selected, or

- The **Structure Diagrams** tool is selected.

General tab
**Depth Sorting**

Both these options should be considered if the Fill Plate option is selected below.

- Sort Geometry option
- Sort Nodes option

**Plates and Slabs**

These options determine the view characteristics of a model with finite elements

- Fill Plates option
- Slab Edges option
- Hide Slab Interior Edges option

**Margin Around Structure**

Sets the percentage of the window to allow around the structure to allow for drawing force diagrams.

Results tab

Sets the results to be displayed and their colors.
Load Case A pull-down list of all defined load cases and combinations.

Forces Check boxes to toggle the display of the particular force on the structure. The color on the adjacent button shows the color of that diagram. Click on the button to select an alternative color.

Increments This sets the number of intermediate results that are shown for each element.

Scales tab
Sets the scales that loads and results are to be displayed to. Note that units for each item is set in the Options dialog. If the range does not provide a suitable scale, try alternative units.

- **Load Scale**: Range 0.001-999999
- **Bending Scale**: Range 0.001-999999
- **Shear Scale**: Range 0.001-999999
- **Axial Scale**: Range 0.001-999999
- **Torsion Scale**: Range 0.001-999999
- **Disp. Scale**: Range 0.001-999999

**Labels tab**

Displays the Labels page of the Diagrams dialog for the active structure window.
The dialog contains check boxes for configuring which labels appear on the structure diagram in the active window.

The user must choose between display of section property References or the names of the Sections.

**Node Labels**
- Nodes Numbers
- Node Points
- Supports

**Element Labels**
- Element Numbers
- Orientation Labels
- Element Length

**Property Labels**
- None (select option)
- Reference (select option)
- Section (select option)

**Member Labels**
- Member Numbers
- Group References

**General**
- Axes Diagram

Force Limits tab
Used to color the stick line model of the structure to indicate which elements have end forces or stresses that fall outside a specified range.

**Label Case**  A pull-down list of all loadcases, combinations and envelopes. Note that loadcase chosen in this pull-down updates the similar pull-down on the Results tab and vice versa.

**View Limits**  Choose the range. Whether the maximum limit, minimum limit or both should remain within range.

**Colour within limit**  Click on the colored box to change the pass color and set the thickness of line in pixels between 1 and 10 (structure elements are always drawn 1 pixel wide).

**Colour outside limit**  Click on the colored box to change the fail color and set the thickness of line in pixels between 1 and 10 (structure elements are always drawn 1 pixel wide).

**OK**  Accepts the changes made and closes the dialog.

**Cancel**  Closes the dialog without making any changes.

**Apply**  Applies any changes made in the dialog. Only active if changes have been made.

**Help**  Opens the STAAD.Pro help window.

**D. Colour Manager** dialog
Used to select graphics colors for various plot elements.

Opens when **View > Colours** (on page 1121) is selected.
**Structure Colours**
Sets the colors used in various highlights.

**Analysis Results**
Sets the colors for the Forces (see also Structure Diagrams) and element Limits.

**Design Results**
Sets the colors for the Pass/Fail ratios and member efficiencies.

**Current Structure Colour Scheme**
Sets the colors for the drawing the wire frame of the structure. The chosen scheme is displayed in the status bar (on page 1217).

**OK**
Accepts the changes made and closes the dialog.

**Cancel**
Closes the dialog without making any changes.

**D. Orientation** dialog
Used to adjust the orientation and projection of the picture of the structure in the active structure window.

Opens when **View > Orientation** (on page 1121) is selected.
Dialog controls

**View**
Choose between Isometric and Perspective drawing.

**Default Orientation To**
Click on one of these buttons to set the Elevation, Distance and Rotation to default values.

**Distance to Structure**
The distance from the viewpoint to the origin of the structure.

**Elevation Angle**
The height above ground level (global $Z = 0$) for Perspective drawing, or the projection angle used for Isometric drawing.

**Rotation Angle**
The rotation of the structure about the global $Z$ axis.

**Apply Immediately**
Check this box to see the picture in the active structure window redraw as soon as changes are made in the dialog.

**Apply**
Click to apply the changes in the dialog to the drawing in the active structure window.
This button is not needed if the Apply Immediately box is checked.

**Restore**
Restores the values in the dialog to those that were current when it was opened. Be careful because any changes made are lost.

**Close**
Closes the dialog and applies any changes made to the picture in the active structure window.
To cancel changes, click on the Restore button before clicking on Close.

D. **Open View** dialog
Used to open a saved view of the structure.

Opens when **View > Open View** (on page 1121) is selected.
Dialog controls

**Views**  Choose the view to be opened from the list of stored views.

**Options**  A new window in which to open the view can be created, or the view can be opened in the currently active structure window.

**OK**  Opens the selected view in the selected window and closes the dialog box.

**Cancel**  Closes the dialog without opening a view.

**Related Links**
- [D. Reopening a saved view](on page 1016)

D. **Add to View** dialog
Used to add elements which are selected in the active window to another view window which has already been created.

Opens when View > View Management > Add to View... (on page 1121) is selected.
### Design

**D. Concrete Design**

<table>
<thead>
<tr>
<th>Structure</th>
<th>The file name of the active structure is displayed.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Views</td>
<td>Select the view name to which the selected elements are to be added.</td>
</tr>
<tr>
<td>OK</td>
<td>Accepts the changes made and closes the dialog.</td>
</tr>
<tr>
<td>Cancel</td>
<td>Closes the dialog without making any changes.</td>
</tr>
</tbody>
</table>

**D. Save View As dialog**

Used to save the view in the active window in the data file for the active structure. The view window can then be closed without losing the view.

Opens when **View > View Management > Save View** (on page 1121) is selected.

![Save View As dialog](image)

**Name**

Specify a name for the new view. A list of existing view names is shown below. Selecting one of these will copy the name into and, once OK is clicked, you will be prompted to over write the existing view.

If the content, diagrams or annotation of a saved view are changed, the view is marked as changed by having an asterisk * added to its name. A view with a * appended to its name must be saved in order to retain the changes.

**Tip:** Because Concrete Design uses an asterisk at the end of a view name to indicate that it has changed since it was last saved, it is not recommended to use an asterisk in a view name.

**OK**

Save the new view and closes the dialog.

**Cancel**

Closes the dialog without saving a new view.

**D. Rename View dialog**

Used to rename saved view in the active window.

Opens when **View > View Management > Rename View** (on page 1121) is selected.
**Old Name**  Displays the existing name of the view selected for renaming.

**New Name**  Specify a new name to use for the selected view.

**OK**  Updates the view name and closes the dialog.

**Cancel**  Closes the dialog without changing the view name.

**D. Toolbar Setup** dialog

Used to open toolboxes.

Opens when:

- **View > Toolbars** (on page 1121) is selected, or
- **Toolbars** is selected from the toolbar right-click pop-up menu.

---

**Dialog controls**

**Toolbars**  Allows you to see a list of all available toolbars in Concrete Design. The toolboxes with a check mark next to them are already open. To open a toolbox, you must place a check mark in its check box by clicking it. Single-clicking a check box or a toolbox's name places a check mark in the check box, which will open the toolbox.

**Large Buttons**  If checked, displays the tool icons as larger than their normal size. The default is unchecked. This feature is useful if Concrete Design is run on a very high resolution screen or if you have difficulty seeing the normal size buttons.

**Note:** Although increasing the button size may help you, it also reduces the space in which Concrete Design can draw the structure.
Show Tool Tips
If checked (the default), the name of the tool appears as you pass the pointer over the tool's icon. This option is helpful for learning names and functions the tools in Concrete Design.

**Tip:** When the mouse cursor is placed over a tool, the Status Bar provides additional context to the tool's function.

OK
Closes the dialog and opens all toolboxes with a check mark next to them in the list box. Tool size and tool tip display will also be applied at this time.

Cancel
Closes the dialog without opening the toolboxes that you just checked or applying other options.

**D. Options** dialog
Used to configure many of the graphical features of Concrete Design.

Opens when **View > Options** (on page 1121) is selected.

**View Highlight** tab
Contains the options to configure the appearance of the highlights between structure windows in Concrete Design.

View Highlight
Changes the view highlight colors of the elements in the active view or in the whole structure window, if it is open. This helps locate elements in the view.

Zoom Area
The zoom area of a Zoom window is highlighted on the original window. It helps locate the zoomed in area on the original.

Width
Edit the width of the highlights using the edit box.
Tables tab

Contains a button to configure the font used in the tables in Concrete Design. Click on the Font button to change the font. A sample of the font chosen is displayed.

Font... Click on the Font button to change the font. A sample of the font chosen is displayed.

Note: This font is only used when Page Control is turned off. This is because with Page Control turned on and a very large font would cause problems on pages that have wide tables and picture views.

Node Labels tab

Contains the options for configuring how the node labels on the picture of the structure appear, if used.

To turn Node Labels on, use the Labels command on the View menu.

Figure 137: Nomenclature used in the placement of node labels
**Style**
Choose the style of the node labels.

**Horizontal Alignment**
Choose an option for the horizontal alignment of the node label with respect to the start point for the text.

**Vertical Alignment**
Choose an option for the vertical alignment of the node label with respect to the start point for the text.

**Pixels Right & Above**
Enter the distance in pixels from the node to the start point for the text.

**Font**
Click on the Font button to change the font used for the label. The font colour can also be changed here.

**Opaque**
Select this option to make the label opaque. This obscures any structure or diagram behind the label, ensuring that it can be read.

**Sample**
A graphical preview of how the current settings will display node labels.

**Element Labels tab**
Contains the options for configuring how the element labels on the picture of the structure appear, if used.

**Note:** To turn Node Labels on, use the Labels command on the View menu.

**Figure 138: Nomenclature used in the placement of element labels**
Elements Style & Properties Style
Choose the style of the labels. Configure the style used for the property independently of the style used for the element reference. The property reference is appended to the element reference separated by a colon. e.g. E34 : P2

Horizontal Alignment
Choose an option for the horizontal alignment of the label with respect to the text start position.

Vertical Alignment
Choose an option for the vertical alignment of the label with respect to the text start position.

Pixels Right & Above
The text start point is positioned with respect to the center point of the element. Enter the distance in pixels from the element center to the start point for the text.

Font
Click on the Font button to change the font used for the label. The font colour can also be changed from here.

Opaque
Check this box to make the label opaque. This obscures any structure or diagram behind the label, ensuring that it can be read.

Angle Text
Check this box to angle the label along the line of the element. If this box is unchecked, the label is drawn horizontally.

Sample
A graphical preview of how the current settings will display element labels.

Plate Labels tab
Structure Units tab

Section Units tab
Force Units tab

**OK** Closes the dialog and applies changes to the graphical display.

**Cancel** Closes the dialog without applying any changes.

**Apply** Applies changes to the graphical display.
Help  Opens the STAAD.Pro help window.

D. Diagrams (slab) dialog
Used to control the graphical display of slab related diagrams and plots.
Opens when View > Diagram (on page 1121) is selected (in the Concrete Slab mode)

General tab
Used to control the view of items pertaining to the slab geometry and constituent elements.

Analysis Contour tab
Used to control the display of the analysis plots.
Diagram
Select which of the effect plots can be generated.

Loading
Sets the active loading envelope or load case.

Spacing of Sample points
Sets the level of plot refinement.

Style
Switches plots to line contour or filled contour.
Principal stress directions will also be shown if the box is selected.

Direction \ Result Set
Sets plot directions with respect to one of the available options.

Design Contour tab
Diagram
Used to select plot content: design moments, required reinforcements or spacing of bars.

Face / Direction / Style
Top or bottom reinforcement together with directions and contour style can also be selected with the available options.

D. Select menu
This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements Cursor</td>
<td>Choosing one of the cursor commands changes the cursor shape and its selection function when it is over a structure window. The elements cursor allows individual analytical elements to be selected and joined into design members.</td>
</tr>
<tr>
<td>Members Cursor</td>
<td>Used for limiting selections to members.</td>
</tr>
<tr>
<td>Plates Cursor</td>
<td>Used for limiting selections to plates.</td>
</tr>
<tr>
<td>Slabs Cursor</td>
<td>Used for limiting selections to slabs.</td>
</tr>
<tr>
<td>Select All</td>
<td>The action of the Select All command depends on the current selection mode and cursor.</td>
</tr>
<tr>
<td></td>
<td>• If the Select Elements cursor is current then all the elements in the active structure window are selected.</td>
</tr>
<tr>
<td></td>
<td>• If the Select Nodes cursor is current then all the nodes in the active structure window are selected.</td>
</tr>
<tr>
<td></td>
<td>• If the Select Element Ends cursor is current then all the element ends in the active structure window are selected. Note that this may look similar to all the elements being selected, but it is not the same.</td>
</tr>
<tr>
<td>By Members</td>
<td>Opens the Select Members dialog (on page 1147), which is used to select members in the active structure window by their member numbers.</td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
</tr>
<tr>
<td>------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>By Section Property</td>
<td>Selects elements in the active structure window according to their section property. Displays the Select by Property dialog which contains a list of the section properties defined in the active structure and the option of 'No Property' (a useful check to see that all elements do have a property prior to analysis). Click on properties in the list to select them. Extend the selection by holding down the CTRL key and clicking on further properties. Click OK to update the selection in the active structure window. To quickly show the Select by Property dialog, click the Select by Property button on one of the toolbars.</td>
</tr>
<tr>
<td>By Design Group</td>
<td>Opens the Select Members by Design Group dialog (on page 1148), which is used to select members in the active view window by entire design group. <strong>Note:</strong> Before selecting by Design Group, at least one design group must be defined.</td>
</tr>
<tr>
<td>New Design Group</td>
<td>Opens the New Design Group dialog (on page 1149), which is used to associate a design brief for a Design Group from the currently selected member(s). A Design Group is the selection of members that are to be designed with one of the Concrete Design modules. <strong>Note:</strong> Before defining a new Design Group, one or more members must be selected in the active structure window.</td>
</tr>
</tbody>
</table>

The menu is available in the modes:

- Design Layer
- Member Design
- Earthquake

D. Select Members dialog
Used to select members in the active structure window by their member numbers.

Opens when **Select > By Member** (on page 1146) is selected.
Displays the Select Members dialog which contains a single list box with all the member numbers. The selection in this list reflects any selection already made on the active structure window.

**Tip:** Extend the selection in the list by holding down the **CTRL** key while selecting new members. To remove a selection from the list, hold down the **CTRL** key and click on the item.

Click **OK** to transfer the selection to the active structure window.

**D. Select By Property** dialog

Used to select elements in the active structure window according to their section property.

Opens when **Select > By Section Property…** (on page 1146) is selected.

**List**

Displays the Select by Property dialog which contains a list of the section properties defined in the active structure and the option of 'No Property' (a useful check to see that all elements do have a property prior to analysis). Click on properties in the list to select them.

**Tip:** Extend the selection by holding down the **CTRL** key and clicking on further properties.

**OK**

Updates the selection in the active structure window.

**Cancel**

Closes the dialog without performing any selections.

**D. Select Members by Design Group** dialog

Used to select members in the active view window by entire design group.

Opens when:

- **Select > By Design Group…** (on page 1146) is selected, or
- **Groups > Select by Design Group…** (on page 1159) is selected.
Displays the Select Design Groups dialog which contains a list of the groups defined in the active structure. Click on groups in the list to select them.

**Note:** Extend the selection by pressing the `<CTRL>` key and clicking on additional groups.

**OK**
Updates the selection in the active structure window and close the dialog.

**Cancel**
Closes the dialog without selecting any members by design group.

**Rename...**
Opens the Rename Design Group dialog (on page 1149).

Rename... is enabled when a single design group is selected in the list.

**Delete**
Click to delete the selected Design Group(s). Only the design group is will be removed; all physical members will remain in the design model.

**Caution:** This action cannot be undone. No confirmation dialog will be presented before deleting groups.

Delete is enabled when one or more design group is selected in the list.

**D. Rename Design Group** dialog
Used to rename an existing design group.

Opens when the Rename... button is clicked in the Select Members by Design Group dialog (on page 1148).

**Old Name**
Displays the existing name of the design group selected for renaming.

**New Name**
Specify a new name to use for the selected design group.

**OK**
Updates the Design Group name and closes the dialog.

**Cancel**
Closes the dialog without changing the group name.

**D. New Design Group** dialog
Used to associate a design brief for a Design Group from the currently selected member(s).

Opens when:

- The Edit Design Group button is clicked on the Design Groups Table, or
- Select > New Design Group (on page 1146) is selected, or
- Groups > New Design Group (on page 1159) is selected, or
- The New Design Group tool is selected.

![New Design Group dialog]

G# Enter a name to identify the design group. The default name is incremented automatically.

Design Brief Select a previously defined group design brief.

OK Creates the Design Group with the specified options and closes the dialog.

Cancel Closes the dialog without creating a Design Group.

### D. Members menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Members Table</td>
<td>Displays the [D. Members table](on page 1151) for the active structure. This table is un-editable, but clicking on a member reference number will highlight that member on the diagram.</td>
</tr>
</tbody>
</table>
### Menu item | What it does
---|---
Form Members | Forms a physical member from a selection of one or more connected elements.
**Note:** One or more analytical elements must be selected for this command to be available.
**Note:** If the selection of elements is not connected then the form member command uses the first connected part of the selection only. A warning regarding this appears.
**Tip:** To view the members in the structure, use the Member Labels option in the Labels dialog which is available from the Labels command on the View menu.
See [D. Design of Members and Slabs](on page 1008) for how to use this tool.
AutoForm Members | Forms a number of Members from a selected range of elements. Each member will obey the general rules for generating a member specified (those listed for *Form Member*), but uses the following rules in addition.
- Vertical elements are converted into columns first.
- Where four elements cross and share a single node, if the elements can form two members, the deeper section will form a continuous member, but the other elements will form parts of two separate members.
Unform Member(s) | The menu is available in the modes:
- Design Layer

D. **Members** table
List all members, including their analytical parts. This table is for information only and is not editable.
Opens when:
- The [Des. Layer | Members page](on page 1202) is selected, or
- When [Members > Members Table](on page 1150) is selected, or
- The [Members Table](tool) tool is selected.
Tip: Selecting a Physical Member (or constituent analytical member) reference number will highlight that member in the View window.

Note: Analytical members are read from the STAAD.Pro input file. See D. Rules for Forming Members (on page 1009) for forming Physical Members.

Table Columns
- **Mem**: The reference number of the member.
- **Emt**: The elements that form the particular member.
- **A B**: The nodes that define the start and end of the element.
- **Prop A, Prop B**: The reference of the section properties at the start and end of the element.
- **Length**: The length of the element (in the chosen units).
- **O. length**: The overall length of the member, reduced by any cutbacks if applied (in the chosen units).

Menu Item | Description | Shortcut
--- | --- | ---
Cut | Copy the selected contents to the clipboard and delete from the table. | <Ctrl+X>
Copy | Copy the selected contents to the clipboard. | <Ctrl+C>
Paste | Paste the clipboard contents to the table. | <Ctrl+V>
Go To... | Opens the dialog box (on page 1120), which is used to specify a row number for easy selection. | <F5>
Other Tables…
Opens the dialog box (on page 1126), which is used to toggle the display of tables in the RC Designer interface.

### D. Slabs menu
This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slabs Table</td>
<td>Opens the [D. Slabs table](on page 1153) for the active structure.</td>
</tr>
<tr>
<td>Form Slab</td>
<td>Forms a structural slab from a selection of one or more connected F.E. elements. Elements must be selected for this command to be available. If the selection of elements is not connected then the form Slab command uses the first connected part of the selection only. A warning regarding this appears. To view the slabs in the structure, use the Slabs Number option in the Labels dialog which is available from the Labels command on the View menu. See [D. Design of Members and Slabs](on page 1008) for how to use this tool.</td>
</tr>
<tr>
<td>AutoForm Slab</td>
<td><strong>Attention:</strong> This feature is currently inactive.</td>
</tr>
<tr>
<td>Uniform Slab</td>
<td>Deletes the selected the slab(s) and free the constituent elements.</td>
</tr>
<tr>
<td>Add to Slab</td>
<td><strong>Attention:</strong> This feature is currently inactive.</td>
</tr>
<tr>
<td>Remove from Slab</td>
<td><strong>Attention:</strong> This feature is currently inactive.</td>
</tr>
</tbody>
</table>

The menu is available in the modes:
- Design Layer
- D. Slabs table
Used to display information for the slabs in the active design file.

**Note:** This table is not editable.
Tip: Selecting a Slab reference number will highlight that slab in the View window.

Table Columns

<table>
<thead>
<tr>
<th>Slab</th>
<th>The reference number of the Slab.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thickness</td>
<td>The slab thickness in cm. If thickness varies, 'Variable' is displayed in this column</td>
</tr>
<tr>
<td>Net Area</td>
<td>The net area of the Slab, excludes area of holes</td>
</tr>
<tr>
<td>Gross Area</td>
<td>The gross area of the Slab</td>
</tr>
<tr>
<td>Holes</td>
<td>Number of holes identified in the Slab</td>
</tr>
<tr>
<td>Plates</td>
<td>Number of F.E. forming the Slab</td>
</tr>
</tbody>
</table>

D. Brief menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Brief</td>
<td>Opens the Edit Design brief dialog for the currently selected design brief. A Design Brief is a set of design parameters, which allows the Concrete Design checks to be done. Each Design Brief is setup for a particular Design Code. Briefs are independent of Member Design Groups so that the same brief can be re-used for several design groups. For example, you may have several groups of beams but only one brief for beams. Alternatively, you may tailor each brief to each member group. See D. Design Briefs (on page 1010) for a list of available codes.</td>
</tr>
<tr>
<td>Design Brief Table</td>
<td>Opens the D. Design Briefs table (on page 1155), which lists the design briefs already defined and the codes of practice used for each brief. See D. Design Briefs (on page 1010) for additional information</td>
</tr>
</tbody>
</table>
The menu is available in the modes:

- Design Layer
- Member Design
- Slab Design
- Earthquake

D. Creating a new design brief

1. Click the first empty Brief cell in the Design Brief table.
2. Type a name for the Brief.
3. Press `<Tab>`.
4. Select a Design code to use for this brief.
5. (Optional) Click the Edit Brief button to specify brief details.

D. Deleting a design brief

1. Select the brief's entry in the Design Brief table.
2. Either:
   - Select Edit > Delete.
   - or
   - Press <Delete>.

D. Editing an existing brief

1. Either:
   - Double click the brief's entry in the Design Brief table.
   - or
   - Select the entry with the cursor and then press the Edit Brief button.
   The `<code>` Design Brief dialog (on page 1218) opens.

D. Changing the design code for a brief

1. Select the code cell for the brief you want to change.
   A drop-down list of all beam and column codes available appears in the cell.
2. Select a new code from the list.

D. Design Briefs table
Lists the design briefs already defined and the codes of practice used for each brief.

Opens when:

- Brief > Design Brief Table is selected, or
- The Des. Layer | Groups/Briefs (on page 1202) page is selected.
Design Brief table

Lists all design briefs in the current model.

- **B#** - Number assigned to the design brief (not editable).
- **Brief** - Title for the design brief. Click in the cell to edit the text.
- **Code** - Design code and type associated with the design brief. Click in the cell to select a different code and type pair.

**Note:** If the brief already has been assigned to a Design Group, then any design data for that group will be removed. A warning dialog will appear asking to confirm a change in this case.

**New Brief**

Opens the **New Design Brief** dialog (on page 1156).

**Edit Brief**

Opens the Edit Design Brief dialog for the selected table entry. Refer to the various codes for details on the design parameters for each.

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;Ctrl+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;Ctrl+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the dialog box (on page 1120), which is used to specify a row number for easy selection.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Other Tables...</td>
<td>Opens the dialog box (on page 1126), which is used to toggle the display of tables in the RC Designer interface.</td>
<td></td>
</tr>
</tbody>
</table>

**D. New Design Brief** dialog

Used to assign a title, design code, and design type to a new brief.

Opens when the New Design Brief button is clicked on the **Design Briefs table** (on page 1155).
B# Specify a title to describe the new design brief.

Design Code Select the governing building code for the design of the members in this brief.

Design Type Select the structural elements used in the group to which this design brief will be assigned: Beam, Column, or Slab.

**Note:** Not all element types are supported for all codes.

OK Creates the new design brief and closes the dialog.

Cancel Closes the dialog without creating a new brief.

D. **Bending Dimensions** dialog

Used to specify custom bending dimension radii for use with a user-specified shear steel yield strength.

Opens when **Bending Dimensions**... is clicked on the Shear Reinforcement tab of a beam design brief dialog.
### Minimum Bending Radius

Enter a minimum bar radius to be used for the corresponding bar size (in current length units).

### Anchorage provided by a bend as a multiple of bar size

(Shear Reinforcement only) Specify a numeric value which represents the anchorage length provided by a bar bend (hook). This value is provided in terms of multiples of a bar diameter, and therefore is unitless.

### OK

Accept the specified values and closes the dialog.
## Cancel

Closes the dialog without updating any value.

## D. Group menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select By Design Group</td>
<td>Opens the Select Members by Design Group dialog (on page 1148), which is used to select members by entire design group. Note: Before selecting by Design Group, member groups must be defined for the structure.</td>
</tr>
<tr>
<td>Group Members List Table</td>
<td>Opens the Group Member List table (on page 1160), which displays a list of the members in the current design group with their Analysis and Design sections.</td>
</tr>
<tr>
<td>New Design Group</td>
<td>Opens the New Design Group dialog (on page 1149), which is used to create a new design group from the currently selected member(s). A Design Group is the selection of members that are to be designed with one of the Concrete Design modules. Note: Before defining a new Design Group, one or more members must be selected in the active structure window.</td>
</tr>
<tr>
<td>Design Group Table</td>
<td>Opens the Design Groups Table (on page 1160), which contains the elements and design briefs associations used for designing elements in groups. This table is opened automatically when the Des.Groups option in Page Control Mode is selected. Tip: To open this table when not running in Page Control Mode, select the table from the Tables dialog (on page 1126).</td>
</tr>
<tr>
<td>Member Summary</td>
<td></td>
</tr>
<tr>
<td>Export Group for Detailing</td>
<td>Opens the Export Design Group to Bentley Rebar dialog (on page 1161), which is used to select members for exporting to external detailing programs, such as Bentley Rebar.</td>
</tr>
</tbody>
</table>

The menu is available in the modes:

- Design Layer
Member Design

D. Group Member List table
Displays the members which make up a Design Group, along with the sections used for analysis and design.
Opens when:

- **Groups > Group Member List Table** is selected, or
- When Edit Design Group button is clicked on the **D. Design Groups table** (on page 1160).

![Group Member List Table](image)

Analysis and Design section displayed are those used in the last analysis or design performed, respectively. The design section will display the last analysis section used if no design has been performed. These cells are for information purposes only and cannot be edited.

**Note:** To add a member to the design group, type the member number, omitting the “M”, in the first column of the table.

See **D. Design Groups** (on page 1012) for information on adding or removing members from a design group.

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;Ctrl+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;Ctrl+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the dialog box (on page 1120), which is used to specify a row number for easy selection.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Other Tables...</td>
<td>Opens the dialog box (on page 1126), which is used to toggle the display of tables in the RC Designer interface.</td>
<td></td>
</tr>
</tbody>
</table>

D. Design Groups table
Contains the elements and design briefs associations used for designing elements in groups. The content of the table is grouped into two tabs: Members and Slabs.
Opens when:

- The Des. Layer | Groups/Briefs mode is selected, or
- when **Groups > Design Group Table** (on page 1159) is selected, or
The Design Groups table tool is selected.

![Image of Design Groups table with Members tab selected]

**Figure 139: Design Groups table with the Members tab selected**

**Members tab**

- **Member design group table**
  Each row of the table represents a separate design group, with the group number, Design Group title, and associated Design Brief.

- **New Design Group**
  Click to open the New Design Group dialog (on page 1149), which is used to create a new design group from selected member(s) and associate it with a design brief.

- **Edit Design Group**
  Click to edit the members included in a design group.

**Slabs tab**

- **Design slab table**
  Each row of the table represents a separate design slab, with the slab number, Design Slab title, and associated Design Brief.

- **New Design Slab**
  Click to open the New Design Slab dialog (on page 1162), which is used to create a new design group from selected slab and associate it with a design brief.

- **Edit Design Slab**
  Click to open the Edit Design Slab dialog, which is used to edit the slab or associated design brief included in a design slab.

**D. Export Group to Bentley Rebar**

Used to select members for exporting to external detailing programs, such as Bentley Rebar. The design data can be saved in either a STAAD Markup Language File (file extension .stml) or a Bentley Rebar Link Binary File (file extension .brl).

Opens when Groups > Export Group for Detailing (on page 1159) is selected.

**Note:** Refer to AD.2006-1004.4.1 in the What's New section of the STAAD.Pro help for additional information on using this feature with Bentley Rebar.
Available list  Contains all members within the selected design group.

Selected list  Contains all members which will be included in the export. By default, all members in the group are included.

Cancel  Closes the dialog without exporting any data.

Export  Closes the dialog and opens a Save As dialog for selecting a location and filename (by default, the convention of [STAAD Input File]_[Group Title] is used for the filename). The file filter is set to either a STAAD Markup Language file or a Bentley Rebar Link binary file, depending on the program to which you want to export the data.

Help  Opens the STAAD.Pro help window.

Note: MicroStation and Bentley Rebar are separate products from STAAD.Pro and are independently licensed. If you do not have copies of these products, please contact your STAAD.Pro provider.

D. New Design Slab dialog
Used to associate a design brief and slab for a Design Slab.

Opens when the New Design Slab button is clicked in the Design Groups Table (on page 1160).
**SD#**
Enter a name to identify the design slab. The default name is incremented automatically.

**Design Brief**
Select a previously defined slab design brief.

**For Slab**
Select a physical slab number.

**OK**
Creates the Design Slab with the specified options and closes the dialog.

**Cancel**
Closes the dialog without creating a Design Slab.

### D. Result Line menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create</td>
<td>Used to define a line through the slab for which a set of force and/or displacement results will be established.</td>
</tr>
<tr>
<td>Delete</td>
<td>Deletes the result line and clears associated result plots.</td>
</tr>
<tr>
<td>Result Line Graph</td>
<td>Opens the <a href="#">Result Line Graph and table</a> (on page 1213), which displays a cross section result of the results displayed on the slab and is used to examine the results.</td>
</tr>
<tr>
<td></td>
<td><strong>Note</strong>: If the graph is already open, then the focus is switched to this window.</td>
</tr>
<tr>
<td></td>
<td>The Result Line Graph displays a cross section result of the results displayed on the Slab. This can be changed by selecting the required result from the toolbar drop list or by selecting View &gt; Diagram.</td>
</tr>
<tr>
<td>Constrain to Slab X</td>
<td>If this option is set, when <a href="#">creating a Result Line</a> (on page 1164), the line is fixed to lie parallel to the Slab X axes.</td>
</tr>
<tr>
<td></td>
<td><strong>Note</strong>: This option can be turned off by selecting Result Line &gt; Unconstrained.</td>
</tr>
</tbody>
</table>
The menu is available in the modes:

- Slab Design

D. Drawing a result line

1. Either:
   
   Select Result Line > Create.
   or
   
   Select the Result Line tool

   The mouse pointer changes to a Result Line cursor.

   **Tip:** When the command is active, a tick is displayed against the menu item and the toolbar button is shown as depressed.

2. Click once to define a start point for the Results Line.
   
   A line is rubber-banded from this point to the mouse pointer.

   **Tip:** Select Result Line > Constrain to Slab X to limit the result line to parallel to the slab local x-axis.

3. Click a second time to define the end point.

   **Note:** While the line can be created by clicking on any two points within the Slab Diagram window, the line will be extended such that it will cross the entire slab and cross any openings if present. See D. Slab Alignment dialog (on page 1165) for more details.

4. Repeat steps 2 through 3 to redraw result lines.

5. Press <Esc> to stop drawing Result Lines (or simply click a different page).

**D. Slab menu**

This menu contains the following commands:
Slab Axes

What it does

Opens the Slab Alignment dialog (on page 1165), which is used to define the X axis of the slab.

Set Axes from Line

What it does

If a result line exists, with this command you can set the slab X axes to coincide with it.

Note: After the slab axes is set, the option Constrain to Slab X is set.

The menu is available in the modes:

- Slab Design

D. Slab Alignment dialog

Used to define the X axis of the slab.

Opens when:

- Slab > Slab Axes (on page 1164) is selected, or
- The Slab Axes tool is selected.

The Slab Alignment dialog can be used to define the X axis of the slab. It contains the following options:

- **Perimeter Node**
  - Used to define a line from one perimeter node to another representing the X axes of the slab. The drop down lists contains all the perimeter nodes.

- **Component Plate**
  - Used to realign the slab axis with the local axis of the selected element from the list.

- **Additional Rotation**
  - This value specifies the further rotation of the slab axis by the specified angle in degrees.

- **OK**
  - Accepts the slab alignment parameters and closes the dialog.

- **Cancel**
  - Closes the dialog without saving any settings.

D. Regions menu

This menu contains the following commands:
### Menu item | What it does
--- | ---
**Draw Boundaries** | Used to draw a line with in a slab, which represents a boundary line for Regions within a slab. It can be drawn from one boundary node to another.

**Select Region Boundaries** | Used to select Region boundaries for deletion. When a Region's boundary is deleted, Concrete Design automatically combines the merged Regions.

**Reset Regions** | This command will erase all Regions boundaries, defaulting back to the original layout that has the slab being a single region.

The menu is available in the modes:

- Slab Design

**Related Links**

- [D. Drawing a new region boundary](#) (on page 1166)
- [D. Deleting a region boundary](#) (on page 1167)
- [D. Changing the title for a region](#) (on page 1168)
- [D. Changing the orientation of a region](#) (on page 1167)

**D. Regions table**

Used to display and edit details for slab design regions.

Opens when the **Concrete Slab | Regions** page is selected.

**Related Links**

- [D. Drawing a new region boundary](#) (on page 1166)
- [D. Deleting a region boundary](#) (on page 1167)
- [D. Changing the title for a region](#) (on page 1168)
- [D. Changing the orientation of a region](#) (on page 1167)

**D. Drawing a new region boundary**

1. Select the **Concrete Slab | Regions** page.
2. Either:
Select **Regions > Draw Boundaries**.

or

Select the **Add Boundary** tool.

The cursor changes to the Add Boundary pointer in the View window.

3. Click the first node to define the boundary between two regions.
   A dashed line is “rubber banded” to this node from the mouse pointer.
4. Click the second node used to define the region boundary.
   A solid line is added to show the boundary between the two newly created regions. Labels and local axis are also added to each region, if these are set in the **Diagram (slab) dialog General tab** (on page 1144).

**Related Links**

- **D. Regions menu** (on page 1165)
- **D. Regions table** (on page 1166)

D. Deleting a region boundary

1. Either:
   Select **Regions > Select Region Boundaries**.
   
   or
   
   Select the **Select Boundary** tool.
   
   The cursor changes to the Select Boundary pointer in the View window.

2. Select the boundary line to be removed.
   The line is highlighted blue.

3. Either:
   Select **Edit > Delete**.
   
   or
   
   Press **Delete**.
   The two regions separated by the boundary line are merged.

**Related Links**

- **D. Regions menu** (on page 1165)
- **D. Regions table** (on page 1166)

D. Changing the orientation of a region

1. Select the **Concrete Slab | Regions** page.
   
   The **Regions** table opens.
   
   2. Select the Orientation cell for the Region you want to reorient.
   
   3. Specify a rotation value for the local x-axis.
Note: The default slab local x-axis is used as the default for each region.

4. Press <Return>.

Related Links
- D. Regions menu (on page 1165)
- D. Regions table (on page 1166)

D. Changing the title for a region

1. Select the Concrete Slab | Regions page.
   The Regions table opens.
2. Select the Name cell for the Region you want to rename.
3. Specify a new name in the cell.
4. Press <Return>.

Related Links
- D. Regions menu (on page 1165)
- D. Regions table (on page 1166)

D. Design menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Options</td>
<td>Opens the Design Options dialog (on page 1168), which is used to control the physical members to be included in a design command.</td>
</tr>
<tr>
<td>Design Now</td>
<td>Designs the current group according to the settings in the Design Options dialog (on page 1168).</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> In the case of Concrete Slab design, the Design &gt; Design menu item designs the selected slab (no options are available).</td>
</tr>
</tbody>
</table>

The menu is available in the modes:

- Member Design
- Slab Design

D. (Members) Design Options dialog

Used to control the physical members to be included in a design command. The current Design Group and associated Design Brief are displayed at the top of the dialog for reference.

Opens when Design > Design Options (on page 1168) is selected.
Available list  Contains all members added to the current Design Group.

Selected list  Members which will be included in the pending design.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the member(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all members from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected member(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all member(s) from the Selected list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

**OK**  Saves the member selection and closes the dialog.

**Cancel**  Closes the dialog without saving changes.

**Design**  Initiates the member design for Selected members.

**Help**  Opens the STAAD.pro help window.
D. Check menu

This menu contains the following commands:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Check Parameters</td>
<td>This command re-opens the Collapse Check Setup dialog (on page 1170) that is displayed when the Earthquake Mode is first entered so that the settings can be redefined. Earthquake page (on page 1216) for more information.</td>
</tr>
</tbody>
</table>

The menu is available in the modes:

- Earthquake Mode

D. Collapse Check Setup dialog

Used to specify seismic parameters for performing a collapse check.

Opens when:

- The Earthquake mode (on page 1216) is selected, or
- Check > Check Parameters... is selected.
**Figure 140: The Collapse Check Setup dialog for ACI 318-05 and EC8-2004 codes, respectively**

**Design Code**
Select the appropriate code governing seismic design: ACI (99 or 2005), Eurocode 8, or TS500.

**National Annex**
(EC8-2004 only) Select the national annex to use with Eurocode 8.

**Ductility Level**
Select the level of ductility for the structure overall.
- ACI 318 and TS 500: Either level 1 or 2.
- EC8: Select either Ductility Class Medium (DCM) or Ductility Class High (DCH).
Earthquake Load case
Select the seismic load case from the list of primary and combination results in the pull-down list.

Check Reinforcement detailing rules for Ductility
(EC8-2004 only) Select this option to.

Curvature ductility factor
(EC8-2004 only)

Check Now
All the joints which have beams and columns connected are checked

Cancel
Closes the dialog without checking for collapse under seismic load.

D. Results menu
This menu contains the following command:

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Results Setup</td>
<td>Opens the Design Results Setup dialog (on page 1173), which is used to control the display of results by design group and color.</td>
</tr>
<tr>
<td>Individual Graphs</td>
<td>Opens a Member Graph window (on page 1175) for the selected element or member. Each window can show results of multiple force types and multiple load cases, combinations and/or envelopes.</td>
</tr>
<tr>
<td>Tip: To change or add load cases or lines to the graph, use the Diagrams command from the View menu.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement Diagram</td>
<td>Displays the result of the design on the current member. The diagram is divided into parts which can be resized by dragging the separator. A Beam Design produces a different diagram to a column design.</td>
</tr>
<tr>
<td>Shear Reinforcement Diagram</td>
<td>Displays the shear result of the design on the current member. The diagram is divided into three parts that can be resized by dragging the separator. A Beam Design produces a different diagram to a column design.</td>
</tr>
<tr>
<td>Plan View</td>
<td>Toggles the view on the top section of a main or shear reinforcement diagram to display the reinforcement from an elevation view to a plan view.</td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the Beam Spans table (on page 1177), which provides member span and support information.</td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the Main Reinforcement table (on page 1179), which shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.</td>
</tr>
<tr>
<td>Menu item</td>
<td>What it does</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Shear Links</td>
<td>Opens the Shear Links table (on page 1182), which shows the shear forces and stresses at the design locations along with the required steel area.</td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the Shear Zones table (on page 1183), which displays the results of the shear design.</td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the Span/Depth Ratio table (on page 1184), which is used to verify the limit of effective depth divided by the actual span length is not exceeded.</td>
</tr>
<tr>
<td>Scheduled Bars</td>
<td>Opens the Member # Scheduled Bars table (on page 1186), which is used to specify individual covers and link sizes for each span, as well as verify the support definitions.</td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the Summary table (on page 1187) for the group design, which displays the pass/fail status of all members in the Design Group.</td>
</tr>
<tr>
<td>Clear Results</td>
<td>Deletes the results of the current design.</td>
</tr>
<tr>
<td>Edit Bar Cage</td>
<td>Opens the M# Bar Positions table (on page 1181), which is used to edit the size and arrangement of reinforcing bars in the column reinforcement bar cage.</td>
</tr>
</tbody>
</table>

The menu is available in the modes:

- Member Design
- Slab Design
- Earthquake

D. Design Results Setup dialog
Used to control the display of results by design group and color.

Opens when Results > Design Results Setup (on page 1172) is selected.
Groups tab

Choose the design groups for which results are to be displayed in the graphics window.

**Available list**  
The Available list shows the groups for which results may be displayed.  
Select a group by clicking on it, then transfer it to the Selected list by clicking on the > button.  
Select more than one at once by holding down the **<CTRL>** key while selecting.  
All the groups in the Available list can be selected by clicking on the >> button.

**Selected list**  
The Selected list shows the groups for which results are displayed.  
To remove a group from the Selected list, highlight it by clicking on it, then click on the < button to return it to the Available list.  
All the groups can be removed from the Selected list by clicking on the << button.

Colour Coding tab

The color coding of the results display can be configured.
Colour
Select the “None” check box and no results are shown on the graphic.

Pass Fail
Select the “Pass/Fail” check box to activate the left hand option list. Select the options to display by picking the check boxes.

To reconfigure the colors used for the various options, pick one of the color blocks next to the option list. The Design Results Pass/Fail Colours dialog opens. Pick the item to be changed from the pull down list, then select the color to be used for displaying this item from the palette displayed.

Select the “Efficiencies” check box to activate the right hand option list.

To reconfigure the colors used for the various options, pick on one of the color blocks next to the option list. The Design Results Efficiency Colors dialog opens. Pick the item to be changed from the pull down list, then select the color to be used for displaying this item from the palette displayed.

OK
Changes the group or color settings and closes the dialog.

Cancel
Closes the dialog without applying any changes.

Apply
 Applies selected changes.

Help
Opens the STAAD.Pro help window.

D. Member Graph window
Show results of multiple force types and multiple load cases, combinations and/or envelopes.

Opens when Results > Individual Graphs (on page 1172) is selected.
Right Click Pop-up Menu

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Copy Picture</td>
<td>Copies the graph to the Windows clipboard, so that it may be pasted into other programs that support pasting of images.</td>
</tr>
<tr>
<td>Take Picture</td>
<td>Opens the Picture # dialog (on page 1120), which is used to save the graph to the photo album.</td>
</tr>
<tr>
<td>Diagrams...</td>
<td>Opens the Diagram (graph) dialog (on page 1176), which is used to control the display of load cases and internal forces displayed in the graph.</td>
</tr>
<tr>
<td>Follow View Selection</td>
<td>When this option is toggled on, the Member Graph updates with the member selection in the active view window.</td>
</tr>
<tr>
<td>Follow Current Member</td>
<td></td>
</tr>
</tbody>
</table>

D. Diagram (graph) dialog

Used to control the display of load cases and internal forces displayed in the graph.

Opens when Diagrams... is selected from the right click pop-up menu in the Member Graph window (on page 1175).
Dialog Box options

**Load Cases list** Select the load cases for which the graphs are to be drawn.

**Tip:** More than one load case can be selected by holding down the CTRL key.

**Colours…** Click on this button to customize the colours of the graphs. This button displays the Graph Colours dialog. Select a graph from the drop-down list and then set its colour by clicking on the palette.

**Forces** Select the forces graphs to be drawn.

**OK** Updates the graph and closes the dialog.

**Cancel** Closes the dialog without making any changes to the graph.

**D. Beam Span table**
Used to specify individual covers and link sizes for each span, as well as verify the support definitions.

Opens when:

- **Results > Beam Spans** is selected, or
- the [Concrete Members | Main Rft. page](on page 1207) is selected

**Spans tab**
Used to setup individual covers and link sizes for each span. The values for the Covers and Link Size are initially set by the values from the Design Brief, but can be edited and set separately for each span.
**Design**

**D. Concrete Design**

![Supports tab](image)

**Supports tab**

Used to define the size and type of support that can be used for shear enhancement. Both the Support Type (No support, Simple or Fixed) and Support Width can be set.

![Table Right-Click Pop-Up Menu](image)

**Table Right-Click Pop-Up Menu**

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;CTRL+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;CTRL+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;CTRL+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the Goto Row dialog (on page 1120), which is used to specify a row number for easy selection.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Menu Item</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the [Beam Spans table](on page 1177), which is used to specify individual covers and link sizes for each span, as well as verify the support definitions.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the [Main Reinforcement table](on page 1179), which shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.</td>
<td></td>
</tr>
<tr>
<td>Shear Links</td>
<td>Opens the [Shear Links table](on page 1182), which shows the shear forces and stresses at the design locations along with the required steel area.</td>
<td></td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the [Shear Zones table](on page 1183), which displays the results of the shear design.</td>
<td></td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the [Span/Depth Ratio table](on page 1184), which is used to verify the limit of effective depth divided by the actual span length is not exceeded.</td>
<td></td>
</tr>
<tr>
<td>Scheduled Bars</td>
<td>Opens the [Scheduled Bars table](on page 1186), which provides a list of all bars sizes, lengths, and bends.</td>
<td></td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the [Summary table](on page 1187) for the design group, which displays the pass/fail status of all members in the Design Group.</td>
<td></td>
</tr>
</tbody>
</table>

**D. Main Reinforcement table**

Shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.

Opens when:

- **Results > Main Reinforcement** (on page 1172) is selected, or
- the Concrete Members | Main Rft. page (on page 1207) is selected

The Hoggging and Sagging tabs display the selected reinforcement under negative and positive bending, respectively.

**Note:** This table is not editable. The values displayed are taken from the design of the current member.
Values that are displayed in green indicate sections where the reinforcement that is provided is to meet the minimum reinforcement requirements, as the design moment is too small. Values that are displayed in red indicate that not all the design requirements were met for the provided reinforcement.

### Table Right-Click Pop-Up Menu

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;CTRL+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;CTRL+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;CTRL+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the Goto Row dialog (on page 1120), which is used to specify a row number for easy selection.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the Beam Spans table (on page 1177), which is used to specify individual covers and link sizes for each span, as well as verify the support definitions.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the Main Reinforcement table (on page 1179), which shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.</td>
<td></td>
</tr>
<tr>
<td>Shear Links</td>
<td>Opens the Shear Links table (on page 1182), which shows the shear forces and stresses at the design locations along with the required steel area.</td>
<td></td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the Shear Zones table (on page 1183), which displays the results of the shear design.</td>
<td></td>
</tr>
</tbody>
</table>
### Design

#### D. Concrete Design

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td>Opens the [Span/Depth Ratio table](on page 1184), which is used to verify the limit of effective depth divided by the actual span length is not exceeded.</td>
<td></td>
</tr>
<tr>
<td>Scheduled Bars</td>
<td>Opens the [Scheduled Bars table](on page 1186), which provides a list of all bars sizes, lengths, and bends.</td>
<td></td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the [Summary table](on page 1187) for the design group, which displays the pass/fail status of all members in the Design Group.</td>
<td></td>
</tr>
</tbody>
</table>

**D. M# Bar Positions table**

Used edit the size and arrangement of reinforcing bars in the column reinforcement bar cage.

Opens when **Results > Edit Bar Cage** (on page 1172) is selected.

![M# Bar Positions Table](image)

The following columns of data are used in each row:

- **Position** - Select the side or top bar position for
- **Dist. to C/L** - Specify the center-line to center-line distance between bars.
- **Size** - Select a bar size to use for the longitudinal bars in this cage.
- **Number** - Number of bars in this cage. This value is not user editable and is defined by the **Position** type.

**Apply** Update the member design with the changes made to the bar positions.

**Reset** Undo all changes made and revert to the program selected arrangements.

**Related Links**

- **D. Adding a new bar cage** (on page 1181)
- **D. Deleting a bar cage** (on page 1182)

**D. Adding a new bar cage**

1. Select **Results > Edit Bar Cage**.
   The **M# Bar Positions table** (on page 1181) opens.
2. Click in the first empty cell in the **Ref** column.
   An incremented bar cage number appears in this cell.
3. Press <Enter> to confirm you want to add a new bar cage. The new bar cage parameters are populated in this table row.

4. Make any changes to the default values required.

5. Click Apply.

Related Links

- D. M# Bar Positions table (on page 1181)

D. Deleting a bar cage

1. Select Results > Edit Bar Cage. The M# Bar Positions table (on page 1181) opens.

2. Select the Ref (row) number of the bar cage you want to delete.

   Note: The first row represents the Corner bars and cannot be deleted or edited.

3. Press <Delete>. The bar cage row is removed from the file.

4. Click Apply.

Related Links

- D. M# Bar Positions table (on page 1181)

D. Shear Links table

Shows the shear forces and stresses at the design locations along with the required steel area.

Opens when Results > Shear Links (on page 1172) is selected.

![Shear Links Table]

Note: This table is not editable. The values displayed are taken from the design of the current member.

Table Right-Click Pop-Up Menu

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;CTRL+X&gt;</td>
</tr>
<tr>
<td>Menu Item</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>----------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>-----------</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;CTRL+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;CTRL+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the Goto Row dialog (on page 1120), which is used to specify a row number for easy selection.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the Beam Spans table (on page 1177), which is used to specify individual covers and link sizes for each span, as well as verify the support definitions.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the Main Reinforcement table (on page 1179), which shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.</td>
<td></td>
</tr>
<tr>
<td>Shear Links</td>
<td>Opens the Shear Links table (on page 1182), which shows the shear forces and stresses at the design locations along with the required steel area.</td>
<td></td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the Shear Zones table (on page 1183), which displays the results of the shear design.</td>
<td></td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the Span/Depth Ratio table (on page 1184), which is used to verify the limit of effective depth divided by the actual span length is not exceeded.</td>
<td></td>
</tr>
<tr>
<td>Scheduled Bars</td>
<td>Opens the Scheduled Bars table (on page 1186), which provides a list of all bars sizes, lengths, and bends.</td>
<td></td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the Summary table (on page 1187) for the design group, which displays the pass/fail status of all members in the Design Group.</td>
<td></td>
</tr>
</tbody>
</table>

**D. Shear Zones table**

Displays the results of the shear design.

The program divides the beams into a number of shear zones where the provided shear reinforcement remains constant.

Opens when Results > Shear Zones (on page 1172) is selected.

![Shear Zones Table](image)

**Note:** This table is not editable. The values displayed are taken from the design of the current member.
Table Right-Click Pop-Up Menu

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;CTRL+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;CTRL+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;CTRL+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the Goto Row dialog (on page 1120), which is used to specify a row</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td></td>
<td>number for easy selection.</td>
<td></td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the Beam Spans table (on page 1177), which is used to specify</td>
<td></td>
</tr>
<tr>
<td></td>
<td>individual covers and link sizes for each span, as well as verify the</td>
<td></td>
</tr>
<tr>
<td></td>
<td>support definitions.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the Main Reinforcement table (on page 1179), which shows the design</td>
<td></td>
</tr>
<tr>
<td></td>
<td>moments, required steel areas and provided steel bars for each location</td>
<td></td>
</tr>
<tr>
<td></td>
<td>along the beam that was checked.</td>
<td></td>
</tr>
<tr>
<td>Shear Links</td>
<td>Opens the Shear Links table (on page 1182), which shows the shear forces</td>
<td></td>
</tr>
<tr>
<td></td>
<td>and stresses at the design locations along with the required steel area.</td>
<td></td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the Shear Zones table (on page 1183), which displays the results of</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the shear design.</td>
<td></td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the Span/Depth Ratio table (on page 1184), which is used to verify</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the limit of effective depth divided by the actual span length is</td>
<td></td>
</tr>
<tr>
<td></td>
<td>not exceeded.</td>
<td></td>
</tr>
<tr>
<td>Scheduled Bars</td>
<td>Opens the Scheduled Bars table (on page 1186), which provides a list of</td>
<td></td>
</tr>
<tr>
<td></td>
<td>all bars sizes, lengths, and bends.</td>
<td></td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the Summary table (on page 1187) for the design group, which displays</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the pass/fail status of all members in the Design Group.</td>
<td></td>
</tr>
</tbody>
</table>

D. Span/Depth Ratio table
Used to verify the limit of effective depth divided by the actual span length is not exceeded.

Opens when Results > Deflection (on page 1172) is selected.
The **Basic Limit** values may be edited here.

**Table Right-Click Pop-Up Menu**

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;CTRL+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;CTRL+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;CTRL+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the <a href="#">Goto Row dialog</a> (on page 1120), which is used to specify a row number for easy selection.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the <a href="#">Beam Spans table</a> (on page 1177), which is used to specify individual covers and link sizes for each span, as well as verify the support definitions.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the <a href="#">Main Reinforcement table</a> (on page 1179), which shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.</td>
<td></td>
</tr>
<tr>
<td>Shear Links</td>
<td>Opens the <a href="#">Shear Links table</a> (on page 1182), which shows the shear forces and stresses at the design locations along with the required steel area.</td>
<td></td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the <a href="#">Shear Zones table</a> (on page 1183), which displays the results of the shear design.</td>
<td></td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the <a href="#">Span/Depth Ratio table</a> (on page 1184), which is used to verify the limit of effective depth divided by the actual span length is not exceeded.</td>
<td></td>
</tr>
<tr>
<td>Scheduled Bars</td>
<td>Opens the <a href="#">Scheduled Bars table</a> (on page 1186), which provides a list of all bars sizes, lengths, and bends.</td>
<td></td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the <a href="#">Summary table</a> (on page 1187) for the design group, which displays the pass/fail status of all members in the Design Group.</td>
<td></td>
</tr>
</tbody>
</table>
**D. Member # Scheduled Bars** table  
Provides a list of all bars sizes, lengths, and bends.

Opens when:

- **Results > Scheduled Bars** (on page 1172) is selected, or
- **Concrete Member | Drawing** (on page 1207) is selected.

![Table image](image.png)

**Note:** This table is not editable. The values displayed are taken from the design of the current member.

### Table Right-Click Pop-Up Menu

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td><code>&lt;CTRL+X&gt;</code></td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td><code>&lt;CTRL+C&gt;</code></td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td><code>&lt;CTRL+V&gt;</code></td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the <strong>Goto Row dialog</strong> (on page 1120), which is used to specify a row number for easy selection.</td>
<td><code>&lt;F5&gt;</code></td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the <strong>Beam Spans table</strong> (on page 1177), which is used to specify individual covers and link sizes for each span, as well as verify the support definitions.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the <strong>Main Reinforcement table</strong> (on page 1179), which shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.</td>
<td></td>
</tr>
<tr>
<td>Shear Links</td>
<td>Opens the <strong>Shear Links table</strong> (on page 1182), which shows the shear forces and stresses at the design locations along with the required steel area.</td>
<td></td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the <strong>Shear Zones table</strong> (on page 1183), which displays the results of the shear design.</td>
<td></td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the <strong>Span/Depth Ratio table</strong> (on page 1184), which is used to verify the limit of effective depth divided by the actual span length is not exceeded.</td>
<td></td>
</tr>
</tbody>
</table>
Design
D. Concrete Design

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scheduled Bars</td>
<td>Opens the [Scheduled Bars table](on page 1186), which provides a list of all bars sizes, lengths, and bends.</td>
<td></td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the [Summary table](on page 1187) for the design group, which displays the pass/fail status of all members in the Design Group.</td>
<td></td>
</tr>
</tbody>
</table>

D. <Design Group> Summary table
Displays the pass/fail status of all members in the Design Group for the current Design Group.

Opens when [Results > Member Summary](on page 1172) is selected.

![EXAMP09 - BS8110 Beam Group - Summary](image)

**Note:** This table is not editable. The values displayed are taken from the design of the current member.

**Tip:** Selecting a row in the Summary table will select the same member in the active View window and any reinforcement diagrams open.

Table Right-Click Pop-Up Menu

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut</td>
<td>Copy the selected contents to the clipboard and delete from the table.</td>
<td>&lt;CTRL+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected contents to the clipboard.</td>
<td>&lt;CTRL+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the clipboard contents to the table.</td>
<td>&lt;CTRL+V&gt;</td>
</tr>
<tr>
<td>Go To...</td>
<td>Opens the [Goto Row dialog](on page 1120), which is used to specify a row number for easy selection.</td>
<td>&lt;F5&gt;</td>
</tr>
<tr>
<td>Beam Spans</td>
<td>Opens the [Beam Spans table](on page 1177), which is used to specify individual covers and link sizes for each span, as well as verify the support definitions.</td>
<td></td>
</tr>
<tr>
<td>Main Reinforcement</td>
<td>Opens the [Main Reinforcement table](on page 1179), which shows the design moments, required steel areas and provided steel bars for each location along the beam that was checked.</td>
<td></td>
</tr>
</tbody>
</table>
### Menu Item

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear Links</td>
<td>Opens the [Shear Links table](on page 1182), which shows the shear forces and stresses at the design locations along with the required steel area.</td>
<td></td>
</tr>
<tr>
<td>Shear Zones</td>
<td>Opens the [Shear Zones table](on page 1183), which displays the results of the shear design.</td>
<td></td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the [Span/Depth Ratio table](on page 1184), which is used to verify the limit of effective depth divided by the actual span length is not exceeded.</td>
<td></td>
</tr>
<tr>
<td>Scheduled Bars</td>
<td>Opens the [Scheduled Bars table](on page 1186), which provides a list of bars sizes, lengths, and bends.</td>
<td></td>
</tr>
<tr>
<td>Member Summary</td>
<td>Opens the [Summary table](on page 1187) for the design group, which displays the pass/fail status of all members in the Design Group.</td>
<td></td>
</tr>
</tbody>
</table>

#### D. Drawing menu

This menu contains the following commands:

The menu is available in the modes:

- Member Design
- Slab Design

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Export to DXF</td>
<td>Opens the [Export to Drawing dialog](on page 1188), which is used to create a DXF file of the current member (please contact Bentley Technical Support for more information on this feature). The file contains elevations, sections and a bending schedule.</td>
</tr>
</tbody>
</table>

**D. Export to Drawing** dialog

Used to create a DXF file of the current member that can be read into the AutoCAD application Multi-RC (please contact Bentley Technical Support for more information on this product). The file contains elevations, sections and a bending schedule.

For DXF output, the concrete outlines, covers, and center lines are all output as simple lines to appropriate layers. The bars are sent as polylines, again to separate layers. The schedule is not exported, but may be printed from the normal report setup.

Opens when:

- **Drawings > Export to DXF** is selected, or
- **The Concrete Member | Drawing page** is selected.
Preview window  Shows how the member will appear when imported into Multi-RC with the current settings.

Size and Scale  This sets the size of paper that the drawing is to be imported into and scale of the drawings to be generated.

Export Format  Select either .DXF or MultiREBAR drawing format.

Title Block  This is used for guidance so that the user can position the drawing without encroaching the space that is provided for title blocks in the final drawing.

Cut Beam after span  When the current beam is a multiple span, then the beam can be cut at support locations and the next section is then continued below.

Spacing  There are three values that are used to locate the drawings onto the drawing sheet. The First Span Left and First Span Top locate the top left corner of the first span relative to the top left corner of the drawing. The value Separation Constant defines the distance between sections of a beam that have been cut using the Cut Section as outlined above.

OK  Closes the dialog and opens a Save As dialog (on page 1190) for specifying a name and location for the drawing file.

Cancel  Closes the dialog without creating a drawing file.
D. Save (Drawing) As dialog
Used to name and select a location when saving a .DXF created from the Export to Drawing dialog.

Controls in this dialog are analogous to common Windows Save As dialogs.

**D. Mode menu**

This menu contains the following commands:

The menu is available in the modes:

- Design Layer
- Member Design
- Slab Design
- Earthquake

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Layer</td>
<td>Switches the operating mode of Concrete Design to the Design Layer mode (on page 1202) and page layout to the Job Information Page.</td>
</tr>
</tbody>
</table>
| Member Design       | This switches the operating mode of Concrete Design to the Concrete Member mode (on page 1207).  
                        | There must be at least one Member Design Group defined to enter this mode. If the current design group is a Member Group, the mode is entered with this group. If the current design group is a Slab Group, then the mode is entered with the first Member Design Group.  
                        | The page layout is determined by the design code associated with the current design group. |
### Menu item | What it does
--- | ---
Slab Design | This switches the operating mode of Concrete Design to the Concrete Slab mode (on page 1212).

There must be at least one Slab Design Group defined to enter this mode. If the current design group is a Slab Group, the mode is entered with this group. If the current design group is a Member Group, then the mode is entered with the first Slab Design Group.

The page layout is determined by the design code associated with the current design group.

Earthquake | This switches the operating mode of Concrete Design to the Earthquake mode (on page 1216).

In this mode, the junctions of all the beams and columns designed with the Concrete Design can be checked to one of a number of design codes.

Page Control | Toggles the Page Control (on page 1202) menu on the left hand side of Concrete Design.

### D. Window menu

This menu contains the following commands:

The menu is available in the modes:

- Design Layer
- Member Design
- Slab Design
- Earthquake

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cascade</td>
<td>Arranges the windows in Concrete Design in an overlapping pattern so that the title bar of each window remains visible.</td>
</tr>
<tr>
<td>Tile Horizontal</td>
<td>Arranges the windows in Concrete Design so that all the windows which are not minimized do not overlap and are as wide as possible. Any minimized window icons are arranged in a gap underneath the tiled windows.</td>
</tr>
<tr>
<td>Tile Vertical</td>
<td>Arranges the windows in Concrete Design so that all the windows which are not minimized do not overlap and are as tall as possible. Any minimized window icons are arranged in a gap underneath the tiled windows.</td>
</tr>
</tbody>
</table>
### Menu item | What it does
--- | ---
Arrange Icons | Arranges the icons of any minimized Concrete Design windows into rows across the bottom of the MDI window. This command does not affect the size or position of any open windows.
Structure Only | Closes all Concrete Design windows for the active structure except for a single whole structure window. If a whole structure window is not currently open, then RC Designer creates a new one.

### D. Help menu
This menu contains the following commands:
The menu is available in the modes:
- Design Layer
- Member Design
- Slab Design
- Earthquake

<table>
<thead>
<tr>
<th>Menu item</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td>Displays the contents page of the Concrete Design Help system. The Help system contains sections for Overview, Engineering Information, Using RC Designer step-by-step, Options and a Command Reference. When looking through Help, the Contents page can be returned to by clicking on the Contents button in the Concrete Design Help window.</td>
</tr>
<tr>
<td>About STAAD.Pro RC</td>
<td>Opens the About STAAD.Pro RC dialog (on page 1192) for the program, which provides the version number and copyright information for the software.</td>
</tr>
</tbody>
</table>

D. About STAAD.Pro RC dialog
Provides the version number and copyright information for the software.
Opens when Help > About is selected.
D. Toolbars

Concrete Design displays two toolbars at the top of the screen by default.

The top toolbar, called the **Standard toolbar** (on page 1193), is consistent. The second toolbar, called the Page/Mode Automatic toolbar, is dependent upon the current page.

A third toolbar, called the **Labels toolbar** (on page 1201), may be turned on, as well.

### D. Standard Toolbar

The Standard toolbox contains icons that enable quick access to commonly used pull-down menu items.

**Note:** View, Zoom, Rotation, and Margin can also be finely controlled using the **Orientation dialog** (on page 1133).

#### Table 82: Tools in the Standard toolbar

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Result</th>
<th>Has the same effect as choosing</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Open" /></td>
<td>Open</td>
<td>Re-opens the current STAAD.Pro input file.</td>
<td>File &gt; Re-open</td>
</tr>
<tr>
<td><img src="image" alt="Save" /></td>
<td>Save</td>
<td>Saves the current open file.</td>
<td>File &gt; Save</td>
</tr>
<tr>
<td><img src="image" alt="Print" /></td>
<td>Print</td>
<td>Opens the <strong>Print dialog</strong> (on page 1114).</td>
<td>File &gt; Print</td>
</tr>
<tr>
<td><img src="image" alt="Print Preview" /></td>
<td>Print Preview</td>
<td>Opens the <strong>Print Preview window</strong> (on page 1115).</td>
<td>File &gt; Print Preview</td>
</tr>
<tr>
<td>Selecting this icon</td>
<td>Result</td>
<td>Has the same effect as choosing</td>
<td>Shortcut</td>
</tr>
<tr>
<td>---------------------</td>
<td>--------</td>
<td>---------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Page Control</td>
<td>Toggles the display of the page control tabs (set to on by default).</td>
<td>Mode &gt; Page Control</td>
<td></td>
</tr>
<tr>
<td>Cut</td>
<td>Deletes the selected object(s) and copies it/them to the clipboard.</td>
<td>Edit &gt; Cut</td>
<td>&lt;Ctrl+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copies the selected object(s) to the clipboard.</td>
<td>Edit &gt; Copy</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
<tr>
<td>Paste</td>
<td>Pastes the contents of the clipboard.</td>
<td>Edit &gt; Paste</td>
<td>&lt;Ctrl+V&gt;</td>
</tr>
<tr>
<td>Undo</td>
<td>Undoes last operation.</td>
<td>Edit &gt; Undo</td>
<td>&lt;Ctrl+Z&gt;</td>
</tr>
<tr>
<td>Zoom</td>
<td></td>
<td>View &gt; Zoom</td>
<td></td>
</tr>
<tr>
<td>Labels</td>
<td></td>
<td>View &gt; Labels</td>
<td></td>
</tr>
<tr>
<td>Diagram</td>
<td></td>
<td>View &gt; Structure Diagrams</td>
<td></td>
</tr>
<tr>
<td>Front View</td>
<td>View the foundation model from the positive Z axis.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>Side View</td>
<td>View the foundation model from the positive X axis.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>Plan View</td>
<td>View the foundation model in plan; from the positive Y axis.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>Isometric View</td>
<td>(Default view) View the model</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>Rotate Up</td>
<td>Rotate the foundation model forward about the X axis.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>Selecting this icon</td>
<td>Result</td>
<td>Has the same effect as choosing</td>
<td></td>
</tr>
<tr>
<td>---------------------</td>
<td>--------</td>
<td>-------------------------------</td>
<td></td>
</tr>
<tr>
<td>![Rotate Down]</td>
<td>Rotate the foundation mode backward about the X axis.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>![Rotate Left]</td>
<td>Rotate the foundation model forward about the Y axis.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>![Rotate Right]</td>
<td>Rotate the foundation model backward about the Y axis.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>![Increase Margin]</td>
<td>Decrease the magnification of the view; thus on increasing the white space around the structure in the view.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
<tr>
<td>![Decrease Margin]</td>
<td>Increase the magnification of the view; thus on reducing the white space around the structure in the view.</td>
<td>View &gt; Orientation</td>
<td></td>
</tr>
</tbody>
</table>

**D. Page/Mode Automatic toolbar**

The contents of this toolbar are dynamically updated based on the current page and mode selection. If the Page Control is not visible, the full toolbar for each mode is displayed. If the Page Control is visible, then only the tools most commonly used on a given page are displayed based on the current page selection.

**D. Design Layer toolbar**

Job Information Page
(No tools used)

Envelopes Page

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Result</th>
<th>Has the same effect as choosing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Current Load Case</td>
<td>Sets the selected load case as current.</td>
<td></td>
</tr>
<tr>
<td>![E1: Envelope 1]</td>
<td>Opens the Define Envelope dialog (on page 1205).</td>
<td></td>
</tr>
</tbody>
</table>
### Members Page

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Result</th>
<th>Has the same effect as choosing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Envelopes Table</td>
<td>Opens the Envelopes table (on page 1204).</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Result</th>
<th>Has the same effect as choosing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select Element Cursor</td>
<td>Select &gt; Elements</td>
<td></td>
</tr>
<tr>
<td>Select Member Cursor</td>
<td>Select &gt; Members</td>
<td></td>
</tr>
<tr>
<td>Form Member</td>
<td>Members &gt; Form Member</td>
<td></td>
</tr>
<tr>
<td>AutoForm Member</td>
<td>Member &gt; Auto Form Members</td>
<td></td>
</tr>
<tr>
<td>Members Table</td>
<td>Opens the Members table (on page 1151).</td>
<td>Members &gt; Members Table</td>
</tr>
<tr>
<td>Node Label</td>
<td>Displays the node numbers in the active View window.</td>
<td>Node Numbers option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
<tr>
<td>Element Label</td>
<td>Displays the element numbers in the active View window.</td>
<td>Element Numbers option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
<tr>
<td>Member Label</td>
<td>Displays the physical member numbers in the active View window.</td>
<td>Member Numbers option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
<tr>
<td>Reference Label</td>
<td>Displays the physical member reference numbers in the active View window.</td>
<td>Properties: References option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
<tr>
<td>Property Name Label</td>
<td>Displays the physical member section properties in the active View window.</td>
<td>Properties: Sections option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
</tbody>
</table>

### Slabs Page
### Selecting this icon | Result | Has the same effect as choosing
--- | --- | ---
Select Plates Cursor | Sets the selected load case as current. | Select > Plates
Select Slabs Cursor | Opens the Define Envelope dialog (on page 1205). | Select > Slabs
Form Slab | | Slabs > Form Slab
Slabs Label | Displays the plate numbers in the active View window. | Slab Numbers option on the Labels tab of the Diagrams dialog (on page 1127).
Plates Label | Displays the slab numbers in the active View window. | Plate Numbers option on the Labels tab of the Diagrams dialog (on page 1127).
Reference Label | Displays the slab reference numbers in the active View window. | Properties: References option on the Labels tab of the Diagrams dialog (on page 1127).
Property Name Label | Displays the slab section properties in the active View window. | Properties: Sections option on the Labels tab of the Diagrams dialog (on page 1127).

### Groups/Briefs Page

| Selecting this icon | Result | Has the same effect as choosing
--- | --- | ---
Select Element Cursor | Select > Elements |  
Select Member Cursor | Select > Members |  
Select Slabs Cursor | Opens the Define Envelope dialog (on page 1205). | Select > Slabs
Current Design Group and Member |  |  
New Design Group | Opens the New Design Group dialog (on page 1149). | Groups > New Design Group
### Design

#### D. Concrete Design

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Result</th>
<th>Has the same effect as choosing</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Members Table" /> Members Table</td>
<td>Opens the Members table (on page 1151).</td>
<td>Members &gt; Members Table</td>
</tr>
<tr>
<td><img src="image" alt="Member Label" /> Member Label</td>
<td>Displays the physical member numbers in the active View window.</td>
<td>Member Numbers option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
<tr>
<td><img src="image" alt="Group Label" /> Group Label</td>
<td>Displays the group number to which a physical member is assigned in the active View window.</td>
<td>Design Group option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
</tbody>
</table>

**D. Concrete Member toolbar**

**Summary, Design, and Drawing Pages**

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Results</th>
<th>Has the same effect as</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Select Element Cursor" /> Select Element Cursor</td>
<td></td>
<td>Select &gt; Elements</td>
</tr>
<tr>
<td><img src="image" alt="Select Member Cursor" /> Select Member Cursor</td>
<td></td>
<td>Select &gt; Members</td>
</tr>
<tr>
<td>Current Design Group and Member</td>
<td><img src="image" alt="G1: beam ▼ ▼" /></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Design Groups Table" /> Design Groups Table</td>
<td>Opens the Design Groups Table (on page 1160).</td>
<td>Groups &gt; Design Groups Table</td>
</tr>
<tr>
<td><img src="image" alt="Members Table" /> Members Table</td>
<td>Opens the Members table (on page 1151).</td>
<td>Members &gt; Members Table</td>
</tr>
<tr>
<td><img src="image" alt="Member Label" /> Member Label</td>
<td>Displays the physical member numbers in the active View window.</td>
<td>Member Numbers option on the Labels tab of the Diagrams dialog (on page 1127).</td>
</tr>
</tbody>
</table>
### Design

**D. Concrete Design**

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Results</th>
<th>Has the same effect as</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Design Brief" /> Design Brief</td>
<td>Opens the <code>&lt;code&gt;</code> &lt;type&gt; Design Brief dialog for the brief associated with the current Design Group. See [D. Design Brief dialogs](on page 1218) for information on the dialogs for a specific member type and code.</td>
<td>Brief &gt; Design Brief</td>
</tr>
</tbody>
</table>

| ![Design Options](image) Design Options | Opens the [Design Options dialog](on page 1168). | Design > Design Options |

Main Layout Page, Main Reinforcement Page, Shear Layout Page and Shear Reinforcement Page

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Results</th>
<th>Has the same effect as</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Select Element Cursor" /></td>
<td></td>
<td>Select &gt; Elements</td>
</tr>
<tr>
<td><img src="image" alt="Select Member Cursor" /></td>
<td></td>
<td>Select &gt; Members</td>
</tr>
<tr>
<td><img src="image" alt="Current Load Case" /></td>
<td>Sets the selected load case as current.</td>
<td></td>
</tr>
</tbody>
</table>

**Individual Graph**

Opens a [Member Graph window](on page 1175) for the selected element or member.

**Force labels, axial, torsion, Y axis moment, Z axis shear, Z axis moment and Y axis shear.**

Toggles the display of the selected force diagram on the modeled elements in the active view window.

| ![Design Groups Table](image) | Opens the [Design Groups Table](on page 1160). | Groups > Design Groups Table |
| ![Beam Spans Table](image) | Opens the [Beam Spans table](on page 1177). | Results > Beam Spans |
| ![Main Reinforcement Table](image) | | Results > Main Reinforcement |
Design
D. Concrete Design

<table>
<thead>
<tr>
<th>Selecting this icon</th>
<th>Results</th>
<th>Has the same effect as</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Shear Link Table]</td>
<td>Results &gt; Shear Link</td>
<td>Shear Link Table</td>
</tr>
<tr>
<td>![Shear Zones Table]</td>
<td>Results &gt; Shear Zones</td>
<td>Shear Zones Table</td>
</tr>
<tr>
<td>![Span/Depth Ratio Table]</td>
<td>Results &gt; Deflection</td>
<td>Span/Depth Ratio Table</td>
</tr>
<tr>
<td>![Shear Legs]</td>
<td>Results &gt; Shear</td>
<td>Shear Legs</td>
</tr>
<tr>
<td>![Anchorage]</td>
<td></td>
<td>Anchorage</td>
</tr>
<tr>
<td>![Plan/ Elevation]</td>
<td>Results &gt; Plan View</td>
<td>Plan/ Elevation</td>
</tr>
</tbody>
</table>

D. Concrete Slab toolbar

Summary Page

<table>
<thead>
<tr>
<th>![SD1: Slab Design 1]</th>
<th>![Bottom : X]</th>
<th>![D. Moment]</th>
</tr>
</thead>
</table>

Group, Direction of Result and Result Type

Displacements and Moments Page

<table>
<thead>
<tr>
<th>![Results Line]</th>
<th></th>
<th></th>
</tr>
</thead>
</table>

Group, Direction of Result and Result Type

Regions Page

<table>
<thead>
<tr>
<th>![Add Boundary]</th>
<th></th>
<th></th>
</tr>
</thead>
</table>

Select Boundary

<table>
<thead>
<tr>
<th>![SD1: Slab Design 1]</th>
<th></th>
<th>Current Design Group</th>
</tr>
</thead>
</table>

Current Design Group

Slab Axes

Design Page

<table>
<thead>
<tr>
<th>![SD1: Slab Design 1]</th>
<th>![Bottom : X]</th>
<th>![D. Moment]</th>
</tr>
</thead>
</table>

Group, Direction of Result and Result Type

D. Earthquake toolbar

Moment Page, Deflection Page, and Shear Pages

<table>
<thead>
<tr>
<th>![Select Elements Cursor]</th>
<th></th>
<th></th>
</tr>
</thead>
</table>
Select Members Cursor

Current Design Group and Member

Node Label

Element Label

Member Label

Design Group Label

**D. Labels toolbar**

Contains tools to toggle the display of labels in the active View window.

*Note:* The Labels toolbar is not shown by default. Select **View > Toolbars** to toggle the display of this toolbar. The labels toolbar and many of the Page Control toolbars have buttons which control individual labels as follows.

<table>
<thead>
<tr>
<th>Icon</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="icon" alt="Node Labels" /></td>
<td>Draws node labels (toggle) in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Node Points" /></td>
<td>Draws a spot (toggle) at the location of the nodes in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Element Labels" /></td>
<td>Draws element labels (toggle) in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Element Length" /></td>
<td>Draws the length of each element (toggle) in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Member Labels" /></td>
<td>Draws member labels (toggle) on elements in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Property Labels" /></td>
<td>Draws section property reference labels (toggle) in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Section Names" /></td>
<td>Draws section name labels (toggle) in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Orientation Symbols" /></td>
<td>Draws orientation symbol labels (toggle) in the active window</td>
</tr>
<tr>
<td><img src="icon" alt="Axes Diagram" /></td>
<td>Draws the axis key diagram (toggle) in the active window</td>
</tr>
</tbody>
</table>
### D. Toolbar Pop-up Menu

Used to quickly toggle the display of toolbars and access toolbar controls.

Opens when a right-click is made on any window toolbar.

<table>
<thead>
<tr>
<th>Menu entry</th>
<th>What it does</th>
<th>Same effect as selecting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard</td>
<td>Toggles the display of the Standard toolbar (on page 1193).</td>
<td></td>
</tr>
<tr>
<td>Page/Mode</td>
<td>Toggles the display of the context sensitive Page/Mode Automatic toolbar (on page 1195).</td>
<td></td>
</tr>
<tr>
<td>Labels</td>
<td>Toggles the display of the Labels toolbar (on page 1201).</td>
<td></td>
</tr>
<tr>
<td>Toolbars...</td>
<td>Opens the Toolbar Setup dialog (on page 1137).</td>
<td>View &gt; Toolbars</td>
</tr>
</tbody>
</table>

### D. Page Control Layout

As with STAAD.Pro, Concrete Design uses a Page Control interface. Each mode has “Pages” that set the layout of the application.

**Tip:** The Page Control display can be toggled by selecting **Mode > Page Control** (on page 1190).

The Page Control menu is comprised of two sets of the buttons, the left side set which indicate the Mode and the right side set which are the Pages. The actual Pages that are displayed are dependent upon the current Mode.

#### D. Design Layer Mode

There are five Pages in the Design Layer. Each page lays out the screen so that you can easily step through the process of setting up and checking data. They are read from the bottom left corner upwards. The pages in the Design Layer are:
<table>
<thead>
<tr>
<th>Page control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Job Information</td>
<td>The Job Information Page is to allow the user to add additional information into the <strong>D. Job Information dialog</strong> (on page 1109). When Concrete Design is first opened, the Job Information dialog is populated with the Job Information entered in STAAD.Pro.</td>
</tr>
<tr>
<td>Envelopes</td>
<td>Used to define envelopes of basic load cases and combination load cases that have been defined in the analysis model. Envelopes are required for Beam Design Briefs. When the Envelopes page is selected, the <strong>D. Envelopes table</strong> (on page 1204) opens in the right of the RC Designer window.</td>
</tr>
<tr>
<td>Members</td>
<td>Designs of beams and columns are performed on physical members which are made of one or more analytical beam parts. The Members Page is to allow the user to combine together a number of individual beam elements and create members. All designs are performed on these members. See <strong>D. Design of Members and Slabs</strong> (on page 1008) for information on how to create a physical member. A list of members, including their analytical parts are listed in the <strong>D. Members table</strong> (on page 1151) on the right of the screen.</td>
</tr>
<tr>
<td>Slabs</td>
<td>Designs of slabs are performed on slab objects which are made of one or more finite element (plate element) parts, which can be later broken down again into design regions. The Slabs Page is to allow the user to combine together a number of individual finite elements and create members. All slab designs are performed on these slab objects. See <strong>D. Design of Members and Slabs</strong> (on page 1008) for information on how to create a slab. A list of all defined slabs, with a summary of their thicknesses, net and gross areas, number of holes and number of plates, are listed in the <strong>Slabs table</strong> (on page 1153) on the right of the screen.</td>
</tr>
</tbody>
</table>
The Groups / Briefs Page is to allow the user to define Design Groups and setup the parameters in Design Briefs. Any number of Design Groups and Design Briefs can be defined, however it is important to note that it is only possible to enter the Design Mode if the security device supports the design code associated with the Design Brief of the current Design Group.

Once one or more Design Groups have been created, the Design Mode (on page 1207) can be entered and the Design Brief of current design group will determine the page layout.

See D. Design Briefs (on page 1010) and D. Design Groups (on page 1012) for additional information on using this page.

D. Envelopes table
Used to manage load envelopes used for element design.

Opens when:

- the Des. Layer | Envelopes page (on page 1202) is selected, or
- The Envelopes table tool is selected.

- Envelopes list Displays the number and title for each envelope used in the current model.
- New Env. Click to open the New Envelopes dialog (on page 1204), which is used to name a new load envelope. Once you have specified a name, the Define Envelopes dialog (on page 1205) will open to define the load cases for the new envelope.
- Edit Env. Opens the Define Envelopes dialog (on page 1205), which is used to modify, rename, or remove load envelopes from the model.
Opens when New Env. is clicked on the Envelopes table (on page 1204).

E# Specify a name for the new load envelope. This will be used to identify the envelope in RC Designer and will be referenced in Reports.

OK Add a new envelope with the specified name and open the Define Envelope dialog (on page 1205).

Cancel Close the dialog without creating a new envelope.

D. Define Envelope dialog
Used to modify, rename, or remove load envelopes from the model.

Opens when:
- the Edit Env. button is clicked on the Envelopes table (on page 1204), or
- when the Define Envelope tool is selected.

*Load Cases*
Displays a list of all basic load cases and load combinations included in the STAAD.Pro input file.

*Envelope*
Select the Envelope number and title from the list of all current envelopes. All included load cases and combinations considered in this envelope is displayed in the list below.
This selection will be set to either a newly created envelope or the current envelope selected in the Design Layer toolbar (on page 1195).

Add basic load cases or combinations to the load envelope using the list transfer tools:

<table>
<thead>
<tr>
<th>Tool</th>
<th>What it Does</th>
<th>Same Effect as</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected item(s) in the Load Case list to the Envelope list.</td>
<td>Double clicking an item in the Load Case list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all currently displayed items in the Load Case list to the Envelope list.</td>
<td></td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all items in the Envelope list.</td>
<td></td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected item(s) in the Envelope list.</td>
<td>Double clicking an item in the Envelope list.</td>
</tr>
</tbody>
</table>

Show Combinations Only
Select this option to limit the Load Cases list to load combinations only (basic load cases will not be displayed).

OK
Applies the changes to the selected Envelope load and closes the dialog.

Cancel
Closes the dialog without making any changes to the load envelope.

Note: If the Define Envelopes dialog was opened through the New Envelope dialog, the newly created envelope is not removed by not loads will be added.

New
Opens the New Envelope dialog (on page 1204), which is used to create a new load envelope.

Rename
Opens the Rename Envelope dialog (on page 1206), which is used to specify a new title for the currently selected envelope.

Delete
Used to remove the currently selected Envelope from the model. A warning dialog opens to verify you want to proceed.

Caution: This action cannot be undone.

D. Rename Envelope dialog
Used to rename an existing load envelope.
Opens when Rename is clicked in the Define Envelope dialog box (on page 1205).
Old Name  Displays the existing name of the envelope selected for renaming.
New Name  Specify a new name to use for the selected envelope.
OK  Updates the envelope name and closes the dialog.
Cancel  Closes the dialog without changing the envelope name.

D. Concrete Member Mode

The layout of the pages of the Concrete Member design mode is dependent on the Design Code associated with the current Design Group.

Concrete Beam pages

A Design Group associated with an Beam Brief will have a Concrete Member Mode containing seven pages. Each page lays out the screen so that the user can easily step through the process of setting up and checking data. They are read from the bottom left corner upwards.

<table>
<thead>
<tr>
<th>Page control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design</td>
<td>Similar in layout to the Summary Page, but on entering the page the Design Options dialog (on page 1168) opens allowing the beam design to be initiated. The &lt;Design Group&gt; Summary table (on page 1187) displays the status of the design.</td>
</tr>
<tr>
<td>Summary</td>
<td>Displays the summary of the design of the current beam Design Group. The page is laid out with a Structure View window on the left and on the right is the Member Summary Table (on page 1187) displaying the status of the members in the current group above the Beam Spans Table (on page 1177) displaying the supports and resulting spans defined for members in the current group.</td>
</tr>
</tbody>
</table>
### Page control

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Main Layout</strong>&lt;br&gt;Graphically displays the main reinforcement resulting from the design of the current beam Design Group.&lt;br&gt;The page is laid out with a Structure View window on the left and a <strong>Main Reinforcement Diagram</strong> of the current member on the right. The current member can be changed by selecting it from the pull-down menu on the second toolbar or by clicking on a member in the current Design Group on the Structure View window.&lt;br&gt;&lt;br&gt;Note: The summary of the reinforcement requirements is displayed in the Main Reinforcement Page.</td>
</tr>
<tr>
<td><strong>Main Reinforcement</strong>&lt;br&gt;Displays the main reinforcement resulting from the design of the current beam Design Group as numerical data.&lt;br&gt;The page is laid out with a Structure View window on the left and on the right the <strong>Beam Spans Table</strong> (on page 1177) of all the members in the current design group, above the <strong>Main Reinforcement Table</strong> (on page 1179) of the current member on the right. The current member can be changed by selecting it from the pull-down menu on the second toolbar or by clicking on a member in the current Design Group on the Structure View window.</td>
</tr>
<tr>
<td><strong>Shear Layout</strong>&lt;br&gt;Displays the shear reinforcement resulting from the design of the current beam Design Group.&lt;br&gt;The page is laid out with a Structure View window on the left and the <strong>Shear Reinforcement Diagram</strong> of the current member on the right. The current member can be changed by selecting it from the menu on the second toolbar or by clicking on a member in the current Design Group on the Structure View window.&lt;br&gt;&lt;br&gt;The <strong>Shear Reinforcement Diagram</strong> displays the shear result of the design on the current member. The diagram is divided into three parts that can be resized by dragging the separator. A Beam Design produces a different diagram to a column design.</td>
</tr>
</tbody>
</table>
### Page control

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear Reinforcement</td>
<td>Displays the shear reinforcement resulting from the design of the current beam Design Group as numerical data. The page is laid out with a Structure View window on the left and on the right the Shear Links Table (on page 1182) above the Shear Zones Table (on page 1183), both for the current member. The current member can be changed by selecting it from the pull-down menu on the second toolbar or by clicking on a member in the current Design Group on the Structure View window.</td>
</tr>
<tr>
<td>Drawing</td>
<td>Used to export member detail drawings and to display a summary of the scheduled bars for the current beam design group. The Bar Schedule Table (on page 1186) opens to the right of the Whole structure View window. This provides a summary of the reinforcing bars used in the current member. When the drawing page is selected, the Export to Drawing dialog (on page 1188) is automatically opened with the current beam information loaded. <strong>Note:</strong> For TS500 beam groups, this tab is labeled Schedule and the Export to Drawing dialog does not open when the page is selected.</td>
</tr>
</tbody>
</table>

### Column Design pages

A Design Group associated with a Column Brief will have a Concrete Member Mode containing seven pages. Each page lays out the screen so that the user can easily step through the process of setting up and checking data. They are read from the bottom left corner upwards.

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Summary</td>
<td>Displays the summary of the design of the current column Design Group. The page is laid out with a Structure View window on the left and on the right is the Member Summary Table (on page 1187) displaying the status of the members in the current group.</td>
</tr>
<tr>
<td>Page control</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| Input        | Displays the design settings of the members in the current column Design Group.  
The page is laid out with a Structure View window on the left and the **Member Design Data table** (on page 1211) on the right. This table is used to override the settings of the design Brief for individual members. The effective length factors, braced directions and cover can be set for each column in the group separately. |
| Design       | Similar to the Summary Page, but on entering the page the **Design Options dialog** (on page 1168) opens allowing the column design to be initiated.  
The **<Design Group> Summary table** (on page 1187) displays the status of the design. |
| Main Layout  | Graphically displays the main reinforcement resulting from the design of the current column Design Group.  
The page is laid out with a Structure View window on the left and a main reinforcement diagram of the current member on the right. The current member can be changed by selecting it from the pull-down menu on the second toolbar or by clicking on a member in the current Design Group on the Structure View window. |
| Shear Layout | Displays the shear reinforcement resulting from the design of the current column Design Group.  
The page is laid out with a Structure View window on the left and a Shear reinforcement diagram of the current member on the right. The current member can be changed by selecting it from the pull-down menu on the second toolbar or by clicking on a member in the current Design Group on the Structure View window. |
| Results      | Displays the reinforcement resulting from the design of the current column Design Group as numerical data.  
The page is laid out with a Structure View window on the left and a Main Reinforcement Table above the Shear Reinforcement Table for the current member on the right. The current member can be changed by selecting it from the pull-down menu on the second toolbar or by clicking on a member in the current Design Group on the Structure View window. |
D. Entering the Concrete Member Design Mode

1. Either:
   
   **Modes > Concrete Member.**
   
   or
   
   the Concrete tab in the Page Control.
   
   The **Concrete Member | Summary** page is displayed.

D. (Column) Member Design Data table

Used to override the settings of the design Brief for individual members. The effective length factors, braced directions and cover can be set for each column in the group separately.

Opens when the **Concrete Member | Input page** (on page 1207) is selected for a column design group.

<table>
<thead>
<tr>
<th>Length</th>
<th>Physical member length as determined by the formed physical member.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Physical member type, assigned by the Design Group associated Design Brief.</td>
</tr>
<tr>
<td>Eff. Len. Factors</td>
<td>The Effective Length factors for major and minor axis bending. These default to the values assigned to the associated design brief. A numeric value can be entered in the cell if a different value is required.</td>
</tr>
</tbody>
</table>
**Design**

**D. Concrete Design**

| **Braced** | The major and minor axis bending braced status is selected using this options. A check is placed if the member is braced in the bending direction. The braced conditions can be changed. |
| **Covers** | The concrete cover dimensions are listed here. The default cover value assigned to the associated design brief are used. A numeric value can be entered if a different value is required. |
| **Link Size** | The size of reinforcing bar is selected from a list here. |

**D. Concrete Slab pages**

The layout of the pages of the Concrete Member design mode is dependent on the Design Code associated with the current Design Group.

Design Codes that are currently available are:

- **BS 8110 Slab** (on page 1260)
- **ACI 318-05/ 318M-05 Slab** (on page 1235)

The engineering information on a BS8110 design can be found in section 3.4.4 (on page 1081)

**Table 83: Sub-pages in the Concrete Slab page**

<table>
<thead>
<tr>
<th>Page name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Summary</td>
<td>Used to view design reinforcement plots, and view tabulated output that lists design moments and bar details with respect to Region axis. Plots can be created for both required reinforcements or bar spacing. Direction settings cover top and bottom reinforcement with respect to local axis of the Region/Slab. You may generate filled or lined plots for all outputs.</td>
</tr>
<tr>
<td>Displacement</td>
<td>Used to evaluate deflection of the slab and produce contour plots. You may draw a Result Line on the plot and view the deflection along the line. The result line can be drawn parallel to the slab axis or in any other direction.</td>
</tr>
<tr>
<td>Moments</td>
<td>Used to interrogate design effects and create plots much the same way as the displacement page.</td>
</tr>
</tbody>
</table>
Regions

Used to divide a Design Slab into Regions, which with its own local axis setting. This page also opens the Regions table (on page 1166). Each Region may represent an area of the slab where you may want to change bar layout.

Note: By default, each ‘Slab’ has one coincident ‘Region’. Designs are carried out for each ‘Region’ and with respect to the Region’s local axis setting.

Regions can be defined in a number of ways. You may draw lines from boundary nodes of elements, or, you may highlight a boundary line and insert a node and the use it for region definition.

Design

Opens the Design Options dialog (on page 1214), used to initiate the design of all regions as per design brief of the slab and local axis of each region.

Related Links

- D. BS8110 Slab Design Principles (on page 1085)

D. Using the Concrete Slab | Displacements page

The Result Line Graph displays a cross section result of the results displayed on the Slab.
A. Select the results to display from this drop-down list or by selecting View > Diagram (on page 1121).

B. A Results Line is drawn onto the slab surface.

**Note:** The graph is set to Filled in the Analysis Contour diagram options (on page 1144).

C. The selected results are displayed along the result line in the Result Line Graph. Numerical results are displayed in the table below.

The Result Line graph and table can also be opened when Result Line > Result Line Graph is selected.

D. Regions table

Used to display and edit details for slab design regions.

Opens when the Concrete Slab | Regions page is selected.

---

**Related Links**

- [D. Drawing a new region boundary](#) (on page 1166)
- [D. Deleting a region boundary](#) (on page 1167)
- [D. Changing the title for a region](#) (on page 1167)
- [D. Changing the orientation of a region](#) (on page 1167)

**D. (Slab) Design Options** dialog

Used to initiate the design of all regions as per design brief of the slab and local axis of each region. Displays all the regions in the active Slab.

Opens when the Concrete Slab | Design page is selected.
**Design**

**D. Concrete Design**

![Design Options dialog](image)

**Note:** All design moments are resolved with respect to local axis of each region before reinforcement calculation proceeds.

- **Regions** Displays the slab regions which will be included in the pending design.
- **OK** Closes the dialog.
- **Cancel** Closes the dialog.
- **Design** Initiates a slab design and closes the dialog.
- **Help** Opens the STAAD.Pro help window.

**D. Entering slab design mode**

1. Either:
   - select **Modes > Slab Design**
   - or
   - select the **Concrete Slab** page.

**D. Changing plot settings**

1. Either:
   - select **View > Diagram**
   - or
   - right click anywhere in the View window and select **Diagram** from the pop-up menu.
The **Diagrams dialog box** (on page 1144) opens with new tabs for slab display.

### D. Earthquake page

When this mode is selected, a **Collapse Check dialog** (on page 1170) is opens. This dialog is used to set the code and code parameters for the seismic load.

#### Table 84: Sub-pages in the Earthquake page

<table>
<thead>
<tr>
<th>Page name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment</td>
<td>Opens the [Moment Collapse Check table](on page 1216).</td>
</tr>
<tr>
<td>Deflection</td>
<td>Opens the [Story Collapse Check table](on page 1216).</td>
</tr>
<tr>
<td>Shear</td>
<td>Opens the [Shear Collapse Check table](on page 1217).</td>
</tr>
<tr>
<td>Reinforcement</td>
<td>Opens the Reinforcement Checks - Beams table and Reinforcement Checks - Columns table.</td>
</tr>
</tbody>
</table>

#### D. Moment Collapse Check table

This feature is not yet documented.

Opens when the [Earthquake | Moment page](on page 1216) is selected.

![EXAMP09 - Moment Collapse Check](image)

#### D. Story Collapse Check table

This feature is not yet documented.

Opens when the [Earthquake | Deflection](on page 1216) page is selected.
D. **Shear Collapse Check** table

This feature is not yet documented.

Opens when the [Earthquake | Shear page](on page 1216) is selected.

D. **Reinforcement Checks - Beams** table

This feature is not yet documented.

D. **Reinforcement Checks - Columns** table

This feature is not yet documented.

D. **Status Bar**

Provides prompts and context-sensitive assistance.
**Selected tool prompt**
Shows a tip for using the tool the mouse pointer is hovering over.

**Colour: <scheme>**
Displays the currently selected color scheme. The color scheme is selected in the Colour Manager dialog (on page 1132).

**Load case title**
Displays the title of the currently selected load case. The load case can be changed using the Load Case selector found in the Page/Mode Automatic toolbar (on page 1195) on various pages.

### D. View Window Controls

The view window contains some view controls in the lower, right corner.

Click E to zoom to the extent of the structure view.

![Zoom controls](image)

Holding the `<CTRL>` key down and pressing one of the direction keys on the keyboard will rotate the structure on the original and zoomed windows.

**Tip:** To quickly open a zoom window, select the Zoom tool on the Standard toolbar.

To move the position of the zoom window use either the scroll bars on the window, the cursor keys on the keyboard, or the Viewpoint tools on the toolbar. The cursor keys and the viewpoint buttons move the viewing point around the structure.

### D. Design Brief dialogs

Used to enter code-specific parameters for a member or slab design brief. The dialog contents is dependent on the code and member type selected.

**D. ACI 318-99 & ACI 318-05 Member Briefs**

**D. ACI 318-99 Beam Brief** dialog
The ACI Beam Design Brief has the following tabs for setting out the design parameters:

- General tab
**Minimum Covers**
Set the cover to the top, bottom and sides of the beams (in inches).

**Concrete**
Set the strength (in ksi), maximum aggregate size (in inches) and option to use lightweight concrete, in which case either the Tensile fct should be defined (in ksi) or the Lightweight Factor as a fraction.

**Forces**
Set the option whether the torsion forces should be designed for and whether the design should be taken at the face or center-line of the support.

**Envelope**
The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

**Divide Beam**
Set the minimum number of segments that each member should be divided into for calculating the reinforcement (must be at least four).

Main Reinforcement tab
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in inches) between the top 2 layers.

Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in inches) between the bottom 2 layers.

Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

Specify the steel strength (in ksi) and whether the steel is epoxy coated.

Select whether the development length is defined from cl 12.2.2 or cl 12.2.3.

Click the Bending Dimensions… button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Note: Bar sizes include: #3, #4, #5, #6, #7, #8, #9, #10, #11, #14, #18

Shear Reinforcement tab
Design Shear For

Select the option to design the shear reinforcement up to the **Center Line of Column** (support), **Column Face** (edge of support), or the distance 'd' from the **Column Face**. Note that the column or support face is defined in the Spans Table.

Select whether to **Include Axial Load Effects** in the longitudinal reinforcement.

Shear Bar Criteria

Specify the Bar Strength to use for shear reinforcement (in ksi).

The bar size to use from the pull-down list of available sizes.

The minimum number of shear legs, 2, 3, 4, 5 or 6.

The minimum spacing between links to consider, if a smaller distance is required, then the number of legs will be increased.

Set whether the stirrups are defined as Closed or Open for the schedule.

Click the **Bending Dimensions...** button to open the **Bending Dimensions dialog** (on page 1157), which is used to specify the bending dimensions for each bar size.

**Note:** Bar sizes include: #3, #4, #5, #6, #7, #8, #9, #10, #11, #14, #18

**OK**  Accepts parameters on all dialog tabs and closes the dialog.

**Cancel**  Closes the dialog without saving any changes.

**Apply**  Saves changes made in the dialog. Only available if changes have been made.

**Help**  Opens the STAAD.Pro help window.

**D. ACI 318-99 Column Design Brief** dialog

The ACI Column Design Brief has the following tabs for setting out the design parameters:
General tab

**Concrete** Specify the strength (in ksi), maximum aggregate size (in inches) and option to use lightweight concrete, in which case either the Tensile fct should be defined (in ksi) or the Lightweight Factor as a fraction.

Specify the cover to the shear links (in inches)

Specify the maximum size of aggregate (in inches)

**Main Bars** Specify the strength of the main bars (in ksi), the range of sizes to consider from minimum to maximum sizes and the bending dimensions.

**Links** Specify the strength of the link bars (in ksi), the minimum size link to consider and the bending dimensions.

For circular columns select whether the design should specify tie or spiral reinforcement.

**Note:** Bar sizes include: #3, #4, #5, #6, #7, #8, #9, #10, #11, #14, #18

Column Parameters tab

**Bracing Conditions** Specify whether the columns are braced in the local y-y and/or z-z directions.

**Effective Length Factors** Specify the effective length factors to be used in the local y-y and z-z directions.

For either direction, whether it is braced (if not specify the stability index to use) and for sidesway whether a specific load case should be considered or the design load case should be used.

**Slenderness Limits** This may be set to 1, or calculated by the design.
Starter bars Specify the area of starter bar steel (in mm$^2$) to ensure that maximum area of steel is not exceeded by the design.

Other Parameters Select whether or not to include design for torsion.
Select whether or not to design as a Sway Frame
Specify the biaxial beta factor.
Specify the beta(d) value to use in the design.

Member Loadcases tab

Used to select load case results which are used for column design with this brief.

Available list Contains all primary load cases and load combinations in the STAAD.Pro input file.

Selected list Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

Related Links
D. ACI 318-05 and 318M-05 Beam Brief dialog
The ACI 318-05 Beam Design Brief has the following tabs for setting out the design parameters:

**Note:** Units used are dependant on the unit system (U.S. Customary or Metric) brief selected. Otherwise, the briefs are identical.

General tab
Design

D. Concrete Design

[Image of ACI 318-05 Beam Design Brief]

- **General**
  - **Minimum Cover (inches)**
    - Top: 2
    - Side: 2
    - Bottom: 2
  - **Concrete**
    - Strength $f_c'$: 4 ksi
    - Aggregate Size: 1.5 in
    - Lightweight: [ ]
      - Tensile fct: 0 ksi
      - Lightweight Factor: 0.75
    - Unit weight: 120 lb/cu ft
  - **Forces**
    - Include Torsion Effects: [ ]
    - At support, take design moment at:
      - Centre Line of Column: [ ]
      - Column Face: [ ]
  - Envelope: Envelope 1
  - Divide Beam into: 12 Segments

[Image of ACI 318M-05 Beam Design Brief]

- **General**
  - **Minimum Cover (millimetres)**
    - Top: 50
    - Side: 50
    - Bottom: 50
  - **Concrete**
    - Strength $f_c'$: 28 MPa
    - Aggregate Size: 40 mm
    - Lightweight: [ ]
      - Tensile fct: 0 MPa
      - Lightweight Factor: 0.75
    - Unit weight: 1920 kg/cu m
  - **Forces**
    - Include Torsion Effects: [ ]
    - At support, take design moment at:
      - Centre Line of Column: [ ]
      - Column Face: [ ]
  - Envelope: Envelope 1
  - Divide Beam into: 12 Segments

[Further images and details not shown]
<table>
<thead>
<tr>
<th><strong>Minimum Cover</strong></th>
<th>Set the cover to the top, bottom and sides of the beams.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Strength fc’</strong></td>
<td>The specified compressive strength of the concrete.</td>
</tr>
<tr>
<td><strong>Aggregate Size</strong></td>
<td>The maximum aggregate size used.</td>
</tr>
</tbody>
</table>
| **Lightweight**        | Select this option if a lightweight aggregate concrete is used. Specify either the average splitting tensile strength of the lightweight concrete value, Tensile fct, (if known) or the Lightweight Factor (by which √fc’ is multiplied, for when Tensile fct is not known) by selecting the appropriate option.  
  The Unit weight of the lightweight concrete must also be specified. |
| **Include Torsion Effects** | Select this option to consider torsion in the design of the reinforcement. |
| **Design moment at support** | Select whether the design section should be taken at the face or center-line of the support. |
| **Envelope**           | The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205). |
| **Divide Beam**        | Set the minimum number of segments that each member should be divided into for calculating the reinforcement (must be at least four). |
Top Bar Criteria

Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.
The Min gap defines the spacing between the top 2 layers.

**Bottom Bar Criteria**

Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing between the bottom 2 layers.

**Side Bar Criteria**

Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

**Main Bar**

Specify the steel strength and whether the steel is epoxy coated.

Select whether the development length is defined from cl 12.2.2 or cl 12.2.3.

Click the **Bending Dimensions**... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

**Note:** Bar sizes include: #3, #4, #5, #6, #7, #8, #9, #10, #11, #14, #18

Shear Reinforcement tab
Design Shear For

Select the option to design the shear reinforcement up to the Center Line of Column (support), Column Face (edge of support), or the distance 'd' from the Column Face. Note that the column or support face is defined in the Spans Table.
Select whether to **Include Axial Load Effects** in the longitudinal reinforcement.

**Shear Bar Criteria**

Specify the Bar Strength to use for shear reinforcement.

The bar size to use from the pull-down list of available sizes.

The minimum number of shear legs, 2, 3, 4, 5 or 6.

The minimum spacing between links to consider, if a smaller distance is required, then the number of legs will be increased.

Set whether the stirrups are defined as **Closed** or **Open** for the schedule.

Click the **Bending Dimensions**… button to open the Bending Dimensions dialog (on page 1157), which is used to specify the bending dimensions for each bar size.

**Note:** Bar sizes include: #3, #4, #5, #6, #7, #8, #9, #10, #11, #14, #18

**OK** Accepts parameters on all dialog tabs and closes the dialog.

**Cancel** Closes the dialog without saving any changes.

**Apply** Saves changes made in the dialog. Only available if changes have been made.

**Help** Opens the STAAD.Pro help window.

**Related Links**

- *D. ACI 318-05 Beam Design Principles* (on page 1096)
- *D. ACI 318-05 Column Design Principles* (on page 1101)
- *D. ACI 318-05 Slab Design Principles* (on page 1104)

**D. ACI 318-05 and 318M-05 Column Design Brief** dialog

The ACI Column Design Brief has the following tabs for setting out the design parameters:

**Note:** Units used are dependant on the unit system (U.S. Customary or Metric) brief selected. Otherwise, the briefs are identical.

Materials tab
### Design

#### D. Concrete Design

**ACI 318-05 Column Design Brief**

- **Concrete**
  - Strength $f'_c$: 4 ksi
  - Aggregate size: 1.5 in
  - Cover: 2 in
  - Lightweight: Off
  - Tensile $f_t$: 0 ksi
  - Lightweight Factor: 0.75
  - Unit weight: 120 lb/cu ft

- **Main Bars**
  - Strength $f_y$: 60 ksi
  - Min Size: #6
  - Max Size: #11

- **Links**
  - Strength $f_y$: 60 ksi
  - Min Size: #4

**ACI 318M-05 Column Brief**

- **Concrete**
  - Strength $f'_c$: 28 MPa
  - Aggregate size: 40 mm
  - Cover: 50 mm
  - Lightweight: Off
  - Tensile $f_t$: 0 MPa
  - Lightweight Factor: 0.75
  - Unit weight: 1920 kg/cu m

- **Main Bars**
  - Strength $f_y$: 420 MPa
  - Min Size: #13
  - Max Size: #35

- **Links**
  - Strength $f_y$: 420 MPa
  - Min Size: #13

For Circular Sections Use:
- Tie Rft.
- Spiral Rft.
**Strength fc’**  The specified compressive strength of the concrete.

**Aggregate Size**  The maximum aggregate size used.

**Cover**  Specify the clear cover used outside the shear links.

**Lightweight**  Select this option if a lightweight aggregate concrete is used. Specify either the average splitting tensile strength of the lightweight concrete value, **Tensile fct** (if known) or the **Lightweight Factor** (by which $\sqrt{fc'}$ is multiplied, for when Tensile fct is not known) by selecting the appropriate option.

The Unit weight of the lightweight concrete must also be specified.

**Main Bars**  Specify the strength of the main bars, the range of sizes to consider from minimum to maximum sizes, and the bending dimensions.

Click the **Bending Dimensions…** button to open the [Bending Dimensions dialog](#) (on page 1157), which is used to specify bending dimensions for each bar size.

**Links**  Specify the strength of the tie or spiral bars (links), the minimum size link to consider, and the bending dimensions.

Click the **Bending Dimensions…** button to open the [Bending Dimensions dialog](#) (on page 1157), which is used to specify bending dimensions for each bar size.

For circular columns select whether the design should specify tie or spiral reinforcement.

**Note:** Bar sizes include: #3, #4,#5, #6, #7, #8, #9, #10, #11, #14, #18

Column Parameters tab
Effective Length Factors
Specify the effective length factors to be used in the local y-y and z-z directions.

Bracing Conditions
Specify whether the columns are braced in the local y-y and/or z-z directions or, if not, the Stability Index Q value for that direction.

Sidesway Load Case
Select whether a specific load case should be considered or the design load case should be used for causing sidesway.
Include Torsion Effects
Select this option to consider torsion in the design of the reinforcement.

Design as Sway Frame
Select this option if the frame(s) which include the column(s) are subject to sidesway.

Starter Bar Area Provided
Specify the area of starter bar steel (in mm²) to ensure that maximum area of steel is not exceeded by the design.

βd
Specify the sustained to total axial load value to use in the design as defined in Chapter 10 of ACI 318. This value is used to calculate the magnified moments for second order effects per section 10.11 and is dependent on the presence of sidesway.

Maximum spacing between main bars
Specify a maximum center-to-center spacing to be used for the longitudinal bars.

Member Loadcases tab
Used to select load case results which are used for column design with this brief.

| Available list | Contains all primary load cases and load combinations in the STAAD.Pro input file. |
| Selected list | Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group. |

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>
## Design

**D. Concrete Design**

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the <strong>Selected</strong> list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

### Related Links
- [D. ACI 318-05 Beam Design Principles](#) (on page 1096)
- [D. ACI 318-05 Column Design Principles](#) (on page 1101)
- [D. ACI 318-05 Slab Design Principles](#) (on page 1104)

**D. ACI 318-05 and 318M-05 Slab Design Brief** dialog
The ACI 318-05 Beam Design Brief has the following tabs for setting out the design parameters:

**Note:** Units used are dependant on the unit system (U.S. Customary or Metric) brief selected. Otherwise, the briefs are identical.

**General tab**
Design

D. Concrete Design

ACI Slab Design Brief

- **General**
  - Minimum Cover:
    - Top: 2 in
    - Bottom: 2 in
  - Concrete:
    - 4 ksi
    - Aggregate Size: 1.5 in
  - Envelope: E1: Envelope 1
  - Minimum distance between sample points: 20 cm

- **Main Reinforcement**

ACI 318M-05 Slab Brief

- **General**
  - Minimum Cover:
    - Top: 50 mm
    - Bottom: 50 mm
  - Concrete:
    - 28 MPa
    - Aggregate Size: 40 mm
  - Envelope: E1: Envelope 1
  - Minimum distance between sample points: 20 cm
**Minimum Cover**
These set the cover to the top and bottom of the slab.

**Concrete Grade**
Specify the compressive concrete strength of the concrete.

**Aggregate Size**
Specify the maximum size of aggregate.

**Envelope**
The slabs will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

**Note:** Top and bottom of slabs are defined by the local axis of the constituent elements.

It may or may not relate to the physical top or bottom of the slab. In order to eliminate any mismatch or confusion, please ensure that all local axes of constituent elements point in the positive y direction of the global coordinate axis.

This ensures consistency of element orientation and alignment of top and bottom of slabs with local axes.

Main Reinforcement tab
Design

There are two ways to carry out slab design. You may set fixed bar spacing and let the program calculate acceptable bar sizes to suit or fix the bar size and calculate acceptable bar spacing.

Main Bar Type

Specify the yield Strength of steel to be used for the main bars either from the available steel grades.

Top/Bottom Bar Criteria

Set the outer bar direction for top reinforcement to follow the ‘X’ or ‘Y’ axis of the slab. Set bar sizes for ‘X’ and ‘Y’ directions. Set bar spacing in the ‘X’ and ‘Y’ direction if design is based on fixed bar size.
**Related Links**
- [D. ACI 318-05 Beam Design Principles](on page 1096)
- [D. ACI 318-05 Column Design Principles](on page 1101)
- [D. ACI 318-05 Slab Design Principles](on page 1104)

**D. AIJ Member Briefs**

**D. AIJ Beam Brief** dialog
The AIJ Beam Design Brief has the following tabs for setting out the design parameters:

**General tab**

- **Minimum cover** Set the cover to the top, bottom and sides of the beams (in mm).
- **Aggregate Size** Specify the maximum size of aggregate to be used (in mm)
- **Concrete** Specify the concrete strength, Fc (in Kg/cm²).
  - Select whether this is a lightweight concrete mix.
- **Loading Type** Select whether the loading is Short Term or Long Term.
- **Envelope** The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the [Define Envelope dialog](on page 1205).
- **Divide Beam** Set the minimum number of segments that each member should be divided into for calculating the reinforcement.
Main Reinforcement tab

**Top Bar Criteria**
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

**Bottom Bar Criteria.**
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

**Main Bar Type.**
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

The available steel grades include: SR235, SRR235, SR295, SRR295, SDR235, SD295A, SD295B, SDR295, SDR345, SD345 and SD390

Specify whether the bars are epoxy coated.

Select whether the 'Development Length' is taken from the ACI 318 code, clause 12.2.2 or 12.2.3

**Side Bar Criteria.**
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

Bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

**Note:** Unless stated otherwise the available bar sizes include: 8, 10, 13, 16, 19, 22, 25, 29, 32, 35, 38, 41

Shear Reinforcement tab
**Design Shear**

Select whether or not to include for the shear forces within the width of the column (or support) by selecting either 'center Line of Support' or 'Column face' (support) or at a distance 'd' from the face of the column (support).

**Shear Bar Criteria.**

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

**Shear Bar Type.**

Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm$^2$) and set the bending dimensions to be used with that strength.

The available steel grades include: SR235, SRR235, SR295, SRR295, SDR235, SD295A, SD295B, SDR295, SDR345, SD345 and SD390

Specify whether the links are 'Open' or 'Closed' for the classifying in the schedule.

**Note:** Unless stated otherwise the available bar sizes include: 8, 10, 13, 16, 19, 22, 25, 29, 32, 35, 38, 41

**Related Links**

- *D. AIJ Beam Design Principles* (on page 1062)

*D. AS 3600 briefs*

*D. AS 3600 Beam Design Brief* dialog

The AS 3600 Beam Design Brief has the following tabs for setting out the design parameters:

General tab
**Minimum covers**  Set the cover to the top, bottom and sides of the beams (in mm).

**Aggregate Size**  Specify the maximum size of aggregate used in the concrete mix (in mm).

**Concrete Grade**  Either select the strength either from the list of available grades in the pull-down list or by defining the strength, $f_{cu}$ (in MPa).

**Envelope**  The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

**Divide Beam**  Set the minimum number of segments that each member should be divided into for calculating the reinforcement.
Top Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The Min gap defines the spacing (in mm) between the top 2 layers.

Bottom Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

Side Bar Criteria
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

Main Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

Shear Reinforcement tab
Design Shear

For

Select whether or not to include for the shear forces within the width of the column (or support) by selecting either Center Line of Support or Column Face (support).

Shear Bar Criteria

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Select if the links are to be detailed as Closed or Open.

Shear Bar Type

Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength.

Click the Bending Dimensions… button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

OK

Accepts parameters on all dialog tabs and closes the dialog.

Cancel

Closes the dialog without saving any changes.

Apply

Saves changes made in the dialog. Only available if changes have been made.

Help

Opens the STAAD.Pro help window.

Related Links

- D. AS 3600 Beam Design Principles (on page 1020)
- D. AS 3600 Column Design Principles (on page 1021)

D. AS 3600 Column Design Brief dialog

The AS 3600 Column Design Brief has the following tabs for setting out the design parameters:

General tab
Concrete Specify the concrete strength either from the available concrete grades or by defining a concrete strength, $f_{cu}$, (in N/mm$^2$).

Specify the minimum center to center spacing for reinforcement.

Specify the cover to the shear reinforcement.

Main Bars Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Links Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.

Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Column Factors tab
Bracing Conditions Specify whether the columns are braced in the local y-y and/or z-z directions.

Effective Length Factors Specify the effective length factors to be used in the local y-y and z-z directions.

Slenderness Limits This may be set to 1 (unity), or calculated by the design.

Area provided by starter bars Specify the area of starter bar steel (in mm²) to ensure that maximum area of steel is not exceeded by the design.

Member Loadcases tab

Used to select load case results which are used for column design with this brief.

Available list Contains all primary load cases and load combinations in the STAAD.Pro input file.
### Selected list
Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

- **OK** Accepts parameters on all dialog tabs and closes the dialog.
- **Cancel** Closes the dialog without saving any changes.
- **Apply** Saves changes made in the dialog. Only available if changes have been made.
- **Help** Opens the STAAD.Pro help window.

**Related Links**
- *D. AS 3600 Beam Design Principles* (on page 1020)
- *D. AS 3600 Column Design Principles* (on page 1021)

### D. BAEL

**D. BAEL 91 Beam Design Brief** dialog
The BAEL Beam Design Brief has the following tabs for setting out the design parameters:

**General tab**
Minimum covers
Set the cover to the top, bottom and sides of the beams (in mm).

High Grip Bond Coefficient
Specify the value of $\psi_s$ (Psi s).

Aggregate Size
Specify the maximum size of aggregate used in the concrete mix (in mm).

Concrete Strength $f_{cj}$
Set the strength either from the list of available grades, or by defining the strength, $f_{cj}$ (in MPa).

Envelope
The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

Divide Beam
Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Main Reinforcement tab
**Top Bar Criteria**  Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

**Bottom Bar Criteria**  Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

**Side Bar Criteria**  Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

**Main Bar Type**  Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

The available steel grades include: Fe E 215, Fe E 235, Fe E 400 and Fe E 500

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Shear Reinforcement tab
Design

Shear For

Select whether or not to include for the shear forces within the width of the column (or support) by selecting either Center Line of Support or Column Face (support).

Select Include Axial Load Effects to account for the axial forces in design of longitudinal reinforcement.

Select whether 'k' in cl. A 5.1.23 should be taken as 0, 1 or calculated from the pull-down list.

Select the option Include Torsional Effects to consider torsion in the selection of reinforcement steel.

Shear Bar Criteria

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Shear Bar Type

Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength.

The available steel grades include: Fe E 215, Fe E 235, Fe E 400 and Fe E 500

Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

SLS Envelope tab
Envelope Select whether any defined envelope should be considered as the serviceability envelope.

Cracking Condition If an envelope is selected, then the cracking condition must be specified as either Detrimental or Very Detrimental.

Safety Factors tab

Concrete $\gamma_b$ Select whether the concrete safety factor, $\gamma_b$ (gamma b), should be taken as 1.5 or 1.0.
**Concrete** θ  Select whether the concrete factor, θ (theta), should be taken as 1.0, 0.9, or 0.85.

**Steel** γ_s  Select whether the steel safety factor, γ_s (gamma s), should be taken as 1.15 or 1.0.

**Related Links**
- *D. BAEL Beam Design Principles* (on page 1050)
- *D. BAEL Column Design Principles* (on page 1051)

**D. BAEL 91 Column Design Brief** dialog

The BAEL Column Design Brief has the following tabs for setting out the design parameters:

**General tab**

- **Concrete** Specify the concrete strength either from the available concrete grades or by defining a concrete strength, fcj, (in MPa).
  - Specify the maximum aggregate size.
  - Specify the cover to the shear reinforcement.
  - Available concrete grades include: 12, 16, 20, 25, 30, 40, 45 and 50

- **Main Bars** Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used (in MPa). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.
  - Specify the minimum and maximum bar sizes to be used.
  - Available steel grades include: Fe E 215, Fe E 235, Fe E 400 and Fe E 500
Links

Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used (in MPa). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.

Available steel grades include: Fe E 215, Fe E 235, Fe E 400 and Fe E 500

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Column Factors tab

Slenderness

Specify the slope of the column e’ as length / (value).

Specify value for α (alpha).

Specify value for φ (phi).

Effective Length Factors

Specify the effective length factors to be used in the local y-y and z-z directions.

Starter bars

Specify the area of starter bar steel (in mm²) to ensure that maximum area of steel is not exceeded by the design.

Cracking Condition

Select whether cracking condition is either: Not Important, Detrimental, or Very Detrimental.

Safety Factors tab
Concrete $\gamma_b$ Select whether the concrete safety factor, $\gamma_b$ (gamma b), should be taken as 1.5 or 1.0.

Steel $\gamma_s$ Select whether the steel safety factor, $\gamma_s$ (gamma s), should be taken as 1.15 or 1.0.

Member Loadcases tab

Used to select load case results which are used for column design with this brief.

Available list Contains all primary load cases and load combinations in the STAAD.Pro input file.

Selected list Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.
<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the <strong>Selected</strong> list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

**Related Links**
- *D. BAEL Beam Design Principles* (on page 1050)
- *D. BAEL Column Design Principles* (on page 1051)

**D. BS8110**

**D. BS 8110 Beam Design Brief** dialog
The BS8110 Beam Design Brief has the following tabs for setting out the design parameters:

**General tab**

<table>
<thead>
<tr>
<th>Minimum Cover</th>
<th>Aggregate Size</th>
<th>Concrete Strength Class</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top 30 mm</td>
<td>Max 20 mm</td>
<td>C40/50</td>
</tr>
<tr>
<td>Side 30 mm</td>
<td>User fcu 50.0 N/mm²</td>
<td></td>
</tr>
<tr>
<td>Bottom 30 mm</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Bond Coefficients**
- Tension: 0.5
- Compression: 0.64

**Envelope**
- Envelope 1
- Divide Beam into 12 Segments

**Minimum covers** Specify the cover to the top, bottom and sides of the beams (in mm).
**Bond Coefficients** Specify the values for use in calculating the anchorage lengths for both tension and compression.

**Aggregate size** Specify the maximum size of aggregate used in the concrete mix (in mm).

**Concrete Grade** Either select a standard grade from the pull-down list or enter the concrete cube strength into the user value (in N/mm$^2$). Available concrete grades include: C30, C35, C40, C45, and C50.

**Envelope** The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

**Divide Beam** Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

**Design for Torsion** Select this option to consider torsion in the design shear and longitudinal reinforcement.

### Main Reinforcement tab

![Main Reinforcement tab](image)

**Top Bar Criteria** Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

**Bottom Bar Criteria** Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

**Side Bar Criteria** Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

**Main Bar Type** Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.
The available steel grades include: T(460) and R(250)

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Shear Reinforcement tab

**Design**

**Shear For**
Select whether or not to include for the shear forces within the width of the column (or support) by selecting either **Center Line of Support** or **Column Face** (support).

Select **Use Enhanced Shear Effects** to increase shear capacity close to column (support) faces as the shear angle increase.

Select **Include Axial Load Effects** to account for the axial forces in design of longitudinal reinforcement.

**Shear Bar Criteria**
Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Select whether the links should be scheduled as **Closed** or **Open**.

**Shear Bar Type**
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength. The available steel grades include: R(250) or T(460)

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

**Related Links**
- *D. BS8110 Beam Design Principles* (on page 1081)
- *D. BS8110 Slab Design Principles* (on page 1085)
- *D. BS8110 Column Design Principles* (on page 1083)

*D. BS 8110 Column Design Brief* dialog
The BS8110 Column Design Brief has the following tabs for setting out the design parameters:

**General tab**

### Concrete
- Specify the concrete strength, $f_{cu}$, either selected as a standard grade or as a user specified value (in N/mm$^2$).
- Available concrete grades include: C30, C35, C40, C45 and C50
- Specify the maximum aggregate size used in the concrete mix (in mm)
- Specify the cover to the shear reinforcement (in mm)

### Main Bars
- Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used (in N/mm$^2$). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.
- Specify the minimum and maximum bar sizes to be used.
- Available steel grades include: T(460) or R(250)

### Links
- Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used (in MPa). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.
- Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.
- Available steel grades include: T(460) or R(250)
**Bracing Conditions**
Specify whether the columns are braced in the local y-y and/or z-z directions.

**Effective Length Factors**
Specify the effective length factors to be used in the local y-y and z-z directions.

**Slenderness Limits**
This may be set to 1, or calculated by the design.

**Area provided by starter bars**
Specify the area of starter bar steel (in mm$^2$) to ensure that maximum area of steel is not exceeded by the design.

**Member Loadcases tab**
Used to select load case results which are used for column design with this brief.
**Available list**  Contains all primary load cases and load combinations in the STAAD.Pro input file.

**Selected list**  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the <em>Available</em> list to the <em>Selected</em> list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the <em>Available</em> list to the <em>Selected</em> list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the <em>Selected</em> list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the <em>Selected</em> list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

**Related Links**
- *D. BS8110 Beam Design Principles* (on page 1081)
- *D. BS8110 Slab Design Principles* (on page 1085)
- *D. BS8110 Column Design Principles* (on page 1083)

**D. BS8110 Slab Design Brief** dialog
The BS8110 Slab Design Brief has the following tabs for setting out the design parameters:

- General tab
**Minimum Cover**

These set the cover to the top and bottom of the slab.

**Aggregate Size**

Aggregate size sets the maximum size of aggregate.

**Concrete Grade**

Either select a standard grade from the pull-down list or enter the concrete cube strength into the user value.

Available concrete grades include: C30, C35, C40, C45 and C50

**Envelope**

The slabs will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

**Note:** Top and bottom of slabs are defined by the local axis of the constituent elements.

It may or may not relate to the physical top or bottom of the slab. In order to eliminate any mismatch or confusion, please ensure that all local axes of constituent elements point in the positive y direction of the global coordinate axis.

This ensures consistency of element orientation and alignment of top and bottom of slabs with local axes.

Main Reinforcement tab
Design

There are two ways to carry out slab design. You may set fixed bar spacing and let the program calculate acceptable bar sizes to suite or, fix the bar size and calculate acceptable bar spacing.

Wood and Armer

Select this option to use Wood & Armer moments.

Main Bar Type

Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used (in N/mm²). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Available steel grades include: T(460) or R(250)

Top/Bottom Bar Criteria.

Set the outer bar direction for top reinforcement to follow the ‘X’ or ‘Y’ axis of the slab.

Set bar sizes for ‘X’ and ‘Y’ directions.

Set bar spacing in the ‘X’ and ‘Y’ direction if design is based on fixed bar size.

Related Links

- D. BS8110 Slab Design Principles (on page 1085)
- D. BS8110 Beam Design Principles (on page 1081)
- D. BS8110 Slab Design Principles (on page 1085)
- D. BS8110 Column Design Principles (on page 1083)

D. CAN/CSA-A23.3-10 Briefs

D. CSA-A23.3-10 Beam Design Brief dialog

The CAN/CSA-A23.3-10 Beam Design Brief has the following tabs for setting out the design parameters:

General tab
Minimum Cover group
Set the cover to the Top, Side, and Bottom of the beams (in mm).

Concrete group
- **Strength f_c'**: Type the design strength of the concrete, f_c' (in MPa).
- **Aggregate Size**: Type the maximum size of aggregate used in the concrete mix (in mm).
- **Density**: Type the density of the concrete used (in kg/m³).
- **Interior / Exterior Exposure**: Select the exposure of the concrete based on its use in the structure.

Forces group
- **Include Torsion Effects**: Set this check box to consider torsion effects on the beam design.
- **At support, take design moment at**: Select where support moments should be considered: Center Line of Column or at Column Face.

Envelope
The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

Divide Beam into Segments
Set the minimum number of segments that each member should be divided into for calculating the reinforcement.
**Dead Load/Total Load Ratio**

Type the ratio of the dead load to total load to consider for service load.

**Main Reinforcement tab**

<table>
<thead>
<tr>
<th><strong>Top Bar Criteria</strong></th>
<th>Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Bottom Bar Criteria</strong></td>
<td>Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.</td>
</tr>
<tr>
<td><strong>Other Bar Criteria</strong></td>
<td>Set either the Minimum Side Bar Size to be used along with minimum and maximum spacing to achieve a suitable spacing on deep beams.</td>
</tr>
</tbody>
</table>
| **Main Bar Group** | **Strength** Type the steel yield strength, \( f_y \), to use for the main reinforcing steel (in MPa).  
**Modulus** Type the modulus of elasticity for the reinforcing steel, \( E_s \) (in MPa).  
**Epoxy Coated** Set this check box when epoxy coated reinforcing steel is used for the main reinforcing. |

Click the **Bending Dimensions...** button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

**Shear Reinforcement tab**

*Note:* Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50
Shear Bar Criteria group

- **Bar Strength**: Type the steel yield strength, \( f_y \), to use for the shear reinforcing steel (in MPa).
- **Min Spacing**: Type a minimum spacing to use between shear bar (in mm).
- **Modulus**: Type the modulus of elasticity for the shear reinforcing steel, \( E_s \) (in MPa).
- **Bar Size**: Select the bar size to use for shear reinforcement.
- **Min No. of Legs**: Select the number of legs (2, 3, 4, 5, or 6) to be used.

Click the **Bending Dimensions...** button to open the **Bending Dimensions dialog** (on page 1157), which is used to specify bending dimensions for each bar size.

**Related Links**
- *D. CAN/CSA-A23.3-10 Beam Design Principles* (on page 1021)
- *D. CSA-A23.3-10 Column Design Brief* dialog
  The CAN/CSA-A23.3-10 Column Design Brief has the following tabs for setting out the design parameters:
  - Materials tab
Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Concrete group
- **Strength $f_c'$**: Type the design strength of the concrete, $f_c'$ (in MPa).
- **Aggregate Size**: Type the maximum size of aggregate used in the concrete mix (in mm).
- **Density**: Type the density of the concrete used (in kg/m$^3$).
- **Cover**: Type the cover to the shear reinforcement (in mm).
- **Interior / Exterior Exposure**: Select the exposure of the concrete based on it's use in the structure.

Main Bars group
- **Strength $f_y$**: Type the steel yield strength, $f_y$, to use for the main reinforcing steel (in MPa).
- **Modulus**: Type the modulus of elasticity for the reinforcing steel, $E_s$ (in MPa).
- **Bending Dimensions**: Click to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.
- **Min / Max Size**: Select the minimum and maximum bar sizes to be used.

Links group
- **Strength $f_{ys}$**: Type the steel yield strength, $f_{ys}$, to use for the shear reinforcing steel (in MPa).
Modulus  Type the modulus of elasticity for the shear reinforcing steel, $E_s$ (in MPa).

Bending Dimensions  Click to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Min Size  Select the minimum bar sizes to be used for the shear links.

For Circular Sections Use  Select either Tie Rft or Spiral Rft to use for confining shear links for circular columns.

### Column Parameters tab

<table>
<thead>
<tr>
<th>Bracing conditions y-y/z-z groups</th>
<th>Effective Length Factor</th>
<th>Type the effective length factor for bending about the indicated axis, $k_y$.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Braced</td>
<td></td>
<td>Set this check box if the columns are braced in the local y-y/z-z axis.</td>
</tr>
<tr>
<td>Stability Index Q</td>
<td></td>
<td>When columns are not Braced, type the stability index, $Q$, for that direction.</td>
</tr>
<tr>
<td>Sidesway Load Case</td>
<td></td>
<td>Use the drop-down list to select the load case to consider for sidesway. The Same as Primary Loadcase option uses the design load case, but any other load case included in the model can be selected.</td>
</tr>
</tbody>
</table>

Include Torsion Effects  Set this option to consider torsion in the design of the reinforcement.

Design as Sway Frame  Set this option if the frames which include the columns are subject to sidesway.
**Design**

D. Concrete Design

---

**Ignore second order effects**

Select this option to ignore magnified moments accounting for second order effects.

**$\beta_d$**

Type the ratio of sustained axial load to total axial load to use in the design as defined in Chapter 10 and clause 10.14.1.3 of CAN/CSA A23.3-04. This value is used to calculate the EI value in equation 10-19.

** Started Bar Area Provided**

Type the area provided for initial main reinforcement steel (in mm$^2$).

**Max spacing between main bars**

Type the maximum spacing to be used between the longitudinal bars (in mm).

---

**Member Loadcases tab**

Used to select load case results which are used for column design with this brief.

---

**Available list**

Contains all primary load cases and load combinations in the STAAD.Pro input file.

**Selected list**

Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

---

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the <strong>Selected</strong> list.</td>
</tr>
</tbody>
</table>
Note: By default, no load cases or combinations are selected.

Related Links
- D. CAN/CSA-A23.3-10 Column Design Principles (on page 1024)

D. CP65

D. CP65 Beam Design Brief dialog

The CP65 Beam Design Brief has the following tabs for setting out the design parameters:

General tab

Minimum covers Specify the cover to the top, bottom and sides of the beams (in mm).

Bond Coefficients Specify the values for use in calculating the anchorage lengths for both tension and compression.

Aggregate size Specify the maximum size of aggregate used in the concrete mix (in mm).

Concrete Grade Either select a standard grade from the pull-down list or enter the concrete cube strength into the user value (in N/mm²).

Envelope The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).
Divide Beam
Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Main Reinforcement tab

Top Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

Bottom Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

Side Bar Criteria
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

Main Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

The available steel grades include: T(460) and R(250)

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Shear Reinforcement tab
Design Shear For
Select whether or not to include for the shear forces within the width of the column (or support) by selecting either 'center Line of Support' or 'Column face' (support).

Select Use enhanced effects to increase shear capacity close to column (support) faces as the shear angle increase.

Select Include Axial Load Effects to account for the axial forces in design of longitudinal reinforcement.

Shear Bar Criteria
Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Shear Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength.

The available steel grades include: R(250) or T(460)

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Related Links
- D. CP65 Beam Design Principles (on page 1072)
- D. CP65 Column Design Principles (on page 1074)

D. CP65 Column Design Brief dialog
The CP65 Column Design Brief has the following tabs for setting out the design parameters:

General tab
Concrete  Specify the concrete strength, $f_{cu}$, either selected as a standard grade or as a user specified value (in N/mm²).

Available concrete grades include: C30, C35, C40, C45 and C50

Specify the maximum aggregate size used in the concrete mix (in mm)

Specify the cover to the shear reinforcement (in mm)

Main Bars  Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used (in N/mm²). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Available steel grades include: T(460) or R(250)

Links  Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used (in MPa). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.

Available steel grades include: T(460) or R(250)
Bracing Conditions
Specify whether the columns are braced in the local y-y and/or z-z directions.

Effective Length Factors
Specify the effective length factors to be used in the local y-y and z-z directions.

Slenderness Limits
This may be set to 1, or calculated by the design.

Starter bars
Specify the area of starter bar steel (in mm$^2$) to ensure that maximum area of steel is not exceeded by the design.

Member Loadcases tab
Used to select load case results which are used for column design with this brief.
Available list  Contains all primary load cases and load combinations in the STAAD.Pro input file.

Selected list  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

Note: By default, no load cases or combinations are selected.

Related Links
- D. CP65 Beam Design Principles (on page 1072)
- D. CP65 Column Design Principles (on page 1074)

D. DIN1045

D. DIN 1045 Beam Design Brief dialog
The DIN 1045-1 Beam Design Brief has the following tabs for setting out the design parameters:

General tab
Top Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

Bottom Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

Side Bar Criteria
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

Envelope
The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

Divide Beam
Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Materials tab
Main Bar Type

Either select a standard grade from the pull-down list or enter the steel yield strength, fyk (in N/mm²) into the user value and set the bending dimensions to be used with that strength as well as a factor to define ftk as a multiple of fy and the ultimate strain.

The available steel grades include: BSt 500 S(A), BSt 500 M(A), BSt 500 S(B) and BSt 500 M(B).

Concrete Grade

Either select a standard grade from the pull-down list or enter the concrete strength, fcu, into the user value (in N/mm²).

If a user strength is defined, then the maximum Elastic Strain and Ultimate Strain must also be defined.

Available concrete grades include: C12/15, C16/20, C20/25, C25/30, C30/37, C35/45, C40/50, C45/55, C50/60, C60/75, C70/85, C80/95, C90/105 and C100/115.

Aggregate Size

Specify the maximum size of aggregate used in the concrete mix (in mm).

Select whether the stress strain relationship should use the Idealised or Slant chart in Ref Bild 27.
Exposition Class

- Select option for **Reinforcement Corrosion, activated by carbonate influences**, either X0, XC1, XC2, XC3 or XC4.
- Select option for **Reinforcement Corrosion, activated by Chlorides, except sea water**, either **Not Applicable**, XD1, XD2 or XD3.
- Select option for **Reinforcement Corrosion, activated by sea water chlorides**, either **Not Applicable**, XS1, XS2 or XS3.
- Select option for **Attack risk by frost with or without de-icing chemicals** either **Not Applicable**, XF1, XF2 or XF3.
- Select option for **Attack risk by chemical attack by the environment** either **Not Applicable**, XA1, XA2 or XA3.
- Select option for **Concrete Attack by abrasion strain**, either 'Not Applicable, XM1, XM2 or XM3.
- Select option for **Fire Resistance Grade as per DIN 4102**, either **F30-B** or **F90-A**
- Specify **Reduction over C(min) for higher grade** (in mm)

The result of these selections will define the concrete cover, C(min), delta(c) and C(nom) and will suggest the minimum concrete grade that will be required for the design.

Shear Reinforcement tab
Design

D. Concrete Design

Design Shear

Select whether the design of shear force is taken to the face of the column (support) by selecting Use shear at support face or Use shear at a distance d from the support face to account for the enhanced shear effects close to the support.

Select option Design for torsion to account for the torsion forces in the beam.

Shear Bar Criteria

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Shear Bar Type

Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength.

The available steel grades include: BSt 500 S(A), BSt 500 M(A), BSt 500 S(B) and BSt 500 M(B)

Note: Unless stated otherwise the available bar sizes include: 6, 8, 10, 12, 14, 16, 20, 25, 28, 32, 35 and 40

Arrangement tab
Specify **The overall depth of section limit for good anchorage bond** (in mm).

Specify **Minimum distance from top surface for good anchorage** (in mm).

Specify **Minimum anchorage length** (in mm)

Specify **Side cover limit relating to alpha(a) coefficient** (in number of diameters)

**Limit for spacing bars**

Set options to include minimum spacing of bars not less than **maximum bar diameter** and/or user dimension (in mm) on top of the limit of \(d(g) + 5\text{mm}\) when \(d(g) > 16\text{mm}\)
### Equation 127, Clause 11.2.2

- Specify **Value for k factor**
- Specify **Value for $f_{ct,eff}$** (in N/mm$^2$)
- Specify **sigma(s)** (as percentage of $f_{yk}$)

Specify **Allowable crack width limit for calculation of maximum allowable spacing** (in mm)

**Related Links**
- *D. German Concrete design per DIN 1045-1* (on page 1052)
- *D. DIN 1045-1 Beam Design Principles* (on page 1052)
- *D. DIN 1045-1 Column Design Principles* (on page 1055)

**D. DIN 1045 Column Design Brief** dialog

The DIN1045-1 Column Design Brief has the following tabs for setting out the design parameters:

- General tab
**Concrete**  Specify the concrete strength, \( f_{ck} \), either selected as a standard grade or as a user specified value (in N/mm\(^2\)).

Available concrete grades include: C12/15, C16/20, C20/25, C25/30, C30/37, C35/45, C40/50, C45/55, C50/60, C60/75, C70/85, C80/95, C90/105 and C100/115

Specify the maximum aggregate size used in the concrete mix (in mm)

Specify the cover to the shear reinforcement (in mm)

**Main Bars**  Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used (in N/mm\(^2\)). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Available steel grades include: BSt 500 S(A), BSt 500 M(A), BSt 500 S(B) and BSt 500 M(B)

**Links**  Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used (in MPa). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 10, 12, 14, 16 or 20.

Available steel grades include: BSt 500 S(A), BSt 500 M(A), BSt 500 S(B) and BSt 500 M(B)

Unless stated otherwise, available bar sizes include: 12, 14, 16, 20, 25, 28, 32, 35 and 40

**Column Factors** tab

---

*STAAD.Pro*  1281  *User Manual*
Slenderness Limits
This may be set to 1, or calculated by the design.

Effective Length Factors
Specify the effective length factors to be used in the local y-y and z-z directions.

Starter bars
Specify the area of starter bar steel (in mm²) to ensure that maximum area of steel is not exceeded by the design.

Specify Maximum spacing between main bars (in mm)
Specify Minimum spacing between link bars (in mm)
Set whether to Use second order theory when slenderness < critical slenderness

User Settings - Main tab
Specify partial safety factors for concrete gamma(c) and steel gamma(s).

Specify sustained compression factors for rectangular and circular columns.

Specify **Minimum** and **Maximum** reinforcement percentages.

Specify **Minimum number of bars to be used in a circular column**.

Specify **Minimum transverse dimension of section** (in mm)

Specify **Upper slenderness limit for model column method**

**Limit for spacing**  Set options to include minimum spacing of bars not less than **maximum bar diameter** bars and/or user dimension (in mm) on top of the limit of $d_g + 5\text{mm}$ when $d_g > 16\text{mm}$

Boxed Values - Shear tab
Specify **Minimum diameter of links** (in mm) and **fraction of diameter of larges longitudinal bar**.

Select options to limit minimum spacing of links from the following:

- **A multiple of the longitudinal bar diameter**, specify the number for multiple.
- **Least dimension of the column**.
- **A minimum distance**, specify the distance (in mm).

**Clause 10.3.3 - Equation - 70**

Specify value of $\sigma_{cd}$.

Specify limit for value of $\rho_1$.

**Member Loadcases tab**

Used to select load case results which are used for column design with this brief.
Available list Contains all primary load cases and load combinations in the STAAD.Pro input file.

Selected list Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td></td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td></td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

Note: By default, no load cases or combinations are selected.

Concrete Stress Block tab
Select option to either calculate the stress block from Table 9 or a set of user values

If the **Custom values** option is selected, then the following must be specified:

- **Ultimate strain limit of concrete** (in millistrain)
- **Elastic strain limit of concrete** (in millistrain)
- **Index of parabolic curve**

**Related Links**
- *D. German Concrete design per DIN 1045-1* (on page 1052)
- *D. DIN 1045-1 Beam Design Principles* (on page 1052)
- *D. DIN 1045-1 Column Design Principles* (on page 1055)

**D. Eurocode 2 2004 briefs**

D. Eurocode 2 2004 Beam Design Brief dialog

The EC2:2004 Beam Design Brief has the following tabs for setting out the design parameters:

- General tab
Minimum covers  Set the cover to the top, bottom and sides of the beams (in mm).
Aggregate Size  Specify the maximum size of aggregate used in the concrete mix (in mm).
Concrete Grade  Either select the strength either from the list of available grades in the pull-down list or by defining the strength, fck (in MPa).
Envelope  The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).
Divide Beam  Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Main Reinforcement tab
Top Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams.
The Min gap defines the spacing (in mm) between the top 2 layers.

Bottom Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams.
The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.
The Min gap defines the spacing (in mm) between the bottom 2 layers.

Side Bar Criteria
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

Main Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.
Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.
Select the Class of the Main Bar from the pull-down list.

Shear Reinforcement tab
**Design Shear For**

Select whether or not to include for the shear forces within the width of the column (or support) by selecting either **Center Line of Support** or **Column Face** (support).

**Shear Bar Criteria**

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Select if the links are to be detailed as **Closed** or **Open**.

**Shear Bar Type**

Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm$^2$) and set the bending dimensions to be used with that strength.

Click the **Bending Dimensions**... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

**National Annex tab**

Used to select a Eurocode 2 National Annex, if necessary, and open a dialog for editing National Annex parameters.
**Country**

Sets the country for which you want to use the National Annex. Select **EC2 without Annex** to use the EC2 code with no national annex changes or additions or to specify custom national annex values.

**Edit button**

Opens a dialog for editing national annex values. The dialog opened is dependant on the selected **Country** option.

- **UK National Annex** - Opens the dialog box (on page 1300), which is used editing EC2 values as directed by the UK National Annex.
- **EC2 without Annex** - Opens the dialog box (on page 1297), which is used to specify general National Annex values.

**Related Links**

- *D. Eurocode 2:2004 Beam Design Principles* (on page 1044)
- *D. Eurocode 2:2004 Column Design Principles* (on page 1047)
- *D. Eurocode 2:2004 Slab Design Principles* (on page 1049)

**D. EC2 2004 Column Design Brief**

The EC2:2004 Column Design Brief has the following tabs for setting out the design parameters:

Materials tab
Concrete

Specify the concrete strength either from the available concrete grades or by defining a concrete strength, fck, (in N/mm²).

Specify the minimum center to center spacing for reinforcement.

Specify the cover to the shear reinforcement.

Include Creep Effects

[BA] Creep effects can be accounted for in the design of compression members as specified in Cl. 5.8.6(3) & (4) of the code. The program requires the Effective Creep Ratio [Cl. 5.8.4(2)] as input to consider the creep effects.

Effective Creep Ratio

If creep is to be accounted for, specify an effective creep ratio to be used.

Main Bars

Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Links

Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.
Click the **Bending Dimensions**… button to open the **Bending Dimensions dialog** (on page 1157), which is used to specify bending dimensions for each bar size.

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

### Column Factors tab

![Diagram of Column Factors tab](image)

- **Inclination**
  - Specify a value for $\theta$, for use in the structural analysis per clause 5.2(5).

- **Slenderness Criterion**
  - Select the slenderness criteria to be used for column design is per clause 5.8.3.1 or clause 5.8.3.3.

- **Ignore second order effects**
  - Select this option to ignore magnified moments accounting for second order effects.

- **Braced Conditions**
  - Specify whether the columns are braced in the local $y$-$y$ and/or $z$-$z$ directions.

- **Effective length factors**
  - Specify the effective length factors to be used in the local $y$-$y$ and $z$-$z$ directions.

- **Starter bar area provided**
  - Specify the area of starter bar steel (in mm^2) to ensure that maximum area of steel is not exceeded by the design.

- **Max spacing between main bars**
  - Specify the maximum spacing to be used between the longitudinal bars.

- **Min spacing between link bars**
  - Specify the minimum spacing to be used between link bars.

### National Annex tab

- Used to select a Eurocode 2 National Annex, if necessary, and open a dialog for editing National Annex parameters.
**Country**  
Sets the country for which you want to use the National Annex. Select **EC2 without Annex** to use the EC2 code with no national annex changes or additions or to specify custom national annex values.

**Edit button**  
Opens a dialog for editing national annex values. The dialog opened is dependant on the selected **Country** option.

- **UK National Annex** - Opens the [dialog box](on page 1300), which is used editing EC2 values as directed by the UK National Annex.
- **EC2 without Annex** - Opens the [dialog box](on page 1297), which is used to specify general National Annex values.

**Member Loadcases tab**  
Used to select load case results which are used for column design with this brief.
Available list Contains all primary load cases and load combinations in the STAAD.Pro input file.

Selected list Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

**Related Links**
- *D. Eurocode 2:2004 Beam Design Principles* (on page 1044)
- *D. Eurocode 2:2004 Column Design Principles* (on page 1047)
- *D. Eurocode 2:2004 Slab Design Principles* (on page 1049)

**D. EC2 2004 Slab Design Brief** dialog
The EC2 2004 Slab Design Brief has the following tabs for setting out the design parameters:

General tab
Minimum Cover
These set the cover to the top and bottom of the slab.

Aggregate Size
Aggregate size sets the maximum size of aggregate.

Concrete Grade
Either select a standard grade from the pull-down list or enter the concrete cube strength into the user value.
Available concrete grades include: C30, C35, C40, C45, and C50

Envelope
The slabs will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

Note: Top and bottom of slabs are defined by the local axis of the constituent elements.

It may or may not relate to the physical top or bottom of the slab. In order to eliminate any mismatch or confusion, please ensure that all local axes of constituent elements point in the positive y direction of the global coordinate axis.

This ensures consistency of element orientation and alignment of top and bottom of slabs with local axes.

Main Reinforcement tab
Design

There are two ways to carry out slab design. You may set fixed bar spacing and let the program calculate acceptable bar sizes to suite or, fix the bar size and calculate acceptable bar spacing.

Main Bar Type

Specify the strength of steel to be used for the main bars either from the available steel grades (S500 or S400) or by specifying the steel yield strength to be used (in N/mm$^2$). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Select the reinforcement Class as either A, B, or C. Reinforcement class is based on Table C.1 of EN 1992-1-1:2004.

Top/Bottom Bar Criteria

Set the outer bar direction for reinforcement to follow the ‘X’ or ‘Y’ axis of the slab.
Set the Bar Size for ‘X’ and ‘Y’ directions.
Set bar Spacing in the ‘X’ and ‘Y’ direction if design is based on fixed bar size.

National Annex tab

Used to select a Eurocode 2 National Annex, if necessary, and open a dialog for editing National Annex parameters.
Country

Sets the country for which you want to use the National Annex. Select **EC2 without Annex** to use the EC2 code with no national annex changes or additions or to specify custom national annex values.

Edit button

Opens a dialog for editing national annex values. The dialog opened is dependant on the selected **Country** option.

- UK National Annex - Opens the dialog box (on page 1300), which is used editing EC2 values as directed by the UK National Annex.
- EC2 without Annex - Opens the dialog box (on page 1297), which is used to specify general National Annex values.

Related Links

- *D. Eurocode 2:2004 Beam Design Principles* (on page 1044)
- *D. Eurocode 2:2004 Column Design Principles* (on page 1047)
- *D. Eurocode 2:2004 Slab Design Principles* (on page 1049)

D. National Annex Values dialog

Used to edit Eurocode 2 2004 concrete design parameters. This allows you to modify values for National Annexes not included in RC Designer by default.

Opens when the Edit button is selected in the **EC2-2004 Beam Design Brief** dialog box (on page 1286) National Annex tab, with the EC2 without annex option selected.

**Save as default values**

Sets the current values on the active dialog box tab as default values.

- **OK** Saves the current values and closes the dialog.
- **Cancel** Closes the dialog without saving any changes.
- **Apply** Saves any changes made in the dialog.
**Partial Factors for Materials 2.4.2.4 (1)**

Specify values to be used as $\text{Gamma } c (\gamma_c)$ and $\text{Gamma } s (\gamma_s)$.

**Design Compressive and Tensile Strengths 3.1.6**

*Alpha cc* is the coefficient to account for the long term effects on the compressive strength of concrete and any unfavorable effects due to the method of application of external loads.

*Alpha ct* is the coefficient to account for the long term effects on the tensile strength of concrete and any unfavorable effects due to the method of application of external loads.

**Slenderness Criterion 5.8.3.1 (1)**

Specify a ratio value (less than 1) which is to satisfy the "predominately" requirement in the note for this clause. This indicates the limit above which the end moments are considered to be caused "predominately" due to imperfections or transverse loads.
Detailing - Section 8 tab

Spacing of Bars 8.2 (2)
Specify values for k1 and k2 factors, which are used to set the minimum spacing requirements for reinforcing bars. The minimum clear bar spacing will be calculated as the maximum of:
- k1 x bar diameter, or
- K2 + maximum aggregate size, or
- 20 mm

Detailing - Section 9 tab

Minimum Tension Steel Area 9.2.1.1 (1)
Specify a coefficient to use in the calculation of $A_{s,\text{min}}$.
Similarly, specify a coefficient for $b_t d$ to take as the minimum value for $A_{s,\text{min}}$. 
### Solid Slab Maximum Bar Spacing 9.3.1.1 (3)
Specify a coefficient to use in the calculation of maximum spacing value to use in slabs (taken as a ratio of h).
Similarly, specify a maximum spacing value in mm.

### Minimum Compression Steel Area 9.5.2 (2)
Specify a coefficient to use in the calculation of $A_{s,\text{min}}$.
Similarly, specify a coefficient for $A_c$ to take as the minimum value for $A_{s,\text{min}}$.

### Maximum Longitudinal Reinforcement 9.5.2 (3)
Specify a coefficient to use in the calculation of $A_{s,\text{min}}$.
Similarly, specify a coefficient for $A_c$ to take as the minimum value for $A_{s,\text{min}}$.

---

#### Shear tab

**C Rd,c**  
Used to calculate the design shear resistance as per equation 6.2.a of the code.

**k1c**  
Used to calculate the design shear resistance as per equation 6.2.a of the code.

**Alpha cw**  
The coefficient to account for the state of the stress in the compression chord of the strut & tie mechanism used to calculate the design shear resistance of the member.

---

**Related Links**
- *D. Eurocode 2:2004 Beam Design Principles* (on page 1044)
- *D. Eurocode 2:2004 Column Design Principles* (on page 1047)
- *D. Eurocode 2:2004 Slab Design Principles* (on page 1049)

---

**D. UK National Annex Values** dialog
Used to control safety factors and other values specified in the UK National Annex to Eurocode 2: 2004.

Opens when the Edit button is selected in the *EC2-2004 Beam Design Brief* dialog box (on page 1286) National Annex tab, with the UK National Annex option selected.
Save as default values  Click to update the default values this dialog with the current values.

Partial Factors for Materials 2.4.2.4 (1) Specify values to be used as Gamma c (γ_c) and Gamma s (γ_s).

Slenderness Criterion 5.8.3.1 (1) Specify a ratio value (less than 1) which is to satisfy the "predominately" requirement in the note for this clause.

OK Saves the current values and closes the dialog.
Cancel Closes the dialog without saving any changes.
Apply Saves any changes made in the dialog.
Help Opens the STAAD.Pro Help window.

Related Links
- D. Eurocode 2:2004 Beam Design Principles (on page 1044)
- D. Eurocode 2:2004 Column Design Principles (on page 1047)
- D. Eurocode 2:2004 Slab Design Principles (on page 1049)

D. ECCS203

D. ECCS 203 Beam Design Brief dialog
The ECCS 203 Beam Design Brief has the following tabs for setting out the design parameters:

General tab
**Minimum cover**  Set the cover to the top, bottom and sides of the beams (in mm).

**Aggregate Size**  Specify the maximum aggregate size used in the concrete mix.

**Bond Coefficients**  Specify the coefficients for both Tension and Compression.

**Concrete Grade**  Set the strength either from the list of available grades, or by defining a cube strength, $f_{cu}$ (in N/mm$^2$).

- Available concrete grades include: C20, C25, C30, C35, C40, C45 or C50

**Envelope**  The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

**Divide Beam**  Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Main Reinforcement tab
### Criteria

<table>
<thead>
<tr>
<th><strong>Top Bar Criteria</strong></th>
<th>Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links. The Min gap defines the spacing (in mm) between the top 2 layers.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Bottom Bar Criteria</strong></td>
<td>Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links. The Min gap defines the spacing (in mm) between the bottom 2 layers.</td>
</tr>
<tr>
<td><strong>Side Bar Criteria</strong></td>
<td>Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.</td>
</tr>
<tr>
<td><strong>Main Bar Type</strong></td>
<td>Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength. The available steel grades include: T(240), T280 and T(360)</td>
</tr>
</tbody>
</table>

| Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50 |

Shear Reinforcement tab
Design Shear For

Select whether or not to include for the shear forces within the width of the column (or support) by selecting either center Line of Support or Column face (support).

Select Use enhanced effects to increase shear capacity close to column (support) faces as the shear angle increase.

Select Include Axial Load Effects to account for the axial forces in design of longitudinal reinforcement.

Shear Bar Criteria

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Shear Bar Type

Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm$^2$) and set the bending dimensions to be used with that strength.

The available steel grades include: T(240), T(280) and T(360)

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Related Links

- [D. ECCS 203 Beam Design Principles](on page 1043)
- [D. ECCS 203 Column Design Principles](on page 1044)

D. ECCS 203 Column Design Brief dialog

The ECCS 203 Column Design Brief has the following tabs for setting out the design parameters:

General tab
### Concrete Grade

Specify the concrete strength either from the available concrete grades or by defining a concrete strength, $f_{cu}$, (in N/mm$^2$).

- Specify the minimum center to center spacing for reinforcement.
- Specify the cover to the shear reinforcement.

Available concrete grades include: C20, C25, C30, C35, C40, C45 and C50

### Main Bars

Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

- Specify the minimum and maximum bar sizes to be used.

Available steel grades include: T(240), T280 and T(360).

### Links

Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

- Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.

Available steel grades include: T(240), T280 and T(360).

Unless stated otherwise, the range of available bars sizes includes: 12, 16, 20, 25, 32 and 40.

---

**Column Factors tab**
Bracing Conditions  Specify whether the columns are braced in the local y-y and/or z-z directions.

Effective Length Factors  Specify the effective length factors to be used in the local y-y and z-z directions.

Slenderness Limits  This may be set to 1, or calculated by the design.

Area provided by starter bars  Specify the area of starter bar steel (in mm$^2$) to ensure that maximum area of steel is not exceeded by the design.

Member Loadcases tab

Used to select load case results which are used for column design with this brief.

Available list  Contains all primary load cases and load combinations in the STAAD.Pro input file.
Selected list

Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

Note: By default, no load cases or combinations are selected.

Related Links
- D. ECCS 203 Beam Design Principles (on page 1043)
- D. ECCS 203 Column Design Principles (on page 1044)

D. EHE

D. EHE Beam Brief
The EHE Beam Design Brief has the following tabs for setting out the design parameters:

General tab
Minimum cover  Set the cover to the top, bottom and sides of the beams (in mm).

Aggregate Size  Maximum size of aggregate to be used (in mm).

Concrete Grade  Set the strength either from the list of available grades, or by defining a cube strength, fck (in N/mm²).

Available concrete grades include: C25, C30, C35, C45 and C50

Envelope  The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

Divide Beam  Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Main Reinforcement tab
Top Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

Bottom Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

Side Bar Criteria
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

Main Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

The available steel grades include: B220S, B400S and B500S

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Shear Reinforcement tab
Design Shear For
Select whether or not to include for the shear forces within the width of the column (or support) by selecting either center Line of Support or Column face (support).

Select Include Axial Load Effects to account for the axial forces in design of longitudinal reinforcement.

Shear Bar Criteria
Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Shear Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength.

The available steel grades include: B220S, B400S and B500S

Boxed Values - General tab

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50
Specify the partial safety factors for concrete, \( \gamma_c \) and steel, \( \gamma_s \).

Specify the **Additional factor for sustained compression** \( \alpha \).

Specify the **Maximum percentage of tension and compression reinforcement**.

Select options for limiting spacing of bars to include:

- **Maximum bar diameter**
- **Specified dimension** (in mm).

Boxed Values - Anchorage & Slender tab

```
<table>
<thead>
<tr>
<th>Partial safety factors</th>
<th>Concrete ( \gamma_c ) 1.50</th>
<th>Steel ( \gamma_s ) 1.15</th>
</tr>
</thead>
<tbody>
<tr>
<td>Additional factor for sustained compression</td>
<td>( \alpha ) 0.85</td>
<td></td>
</tr>
<tr>
<td>Maximum percentage of tension and compression reinforcement</td>
<td>4.00 %</td>
<td></td>
</tr>
</tbody>
</table>
```

Spacing of bars is not less than:

- \( d_b + 5 \text{ mm when } d_b > 32 \text{ mm} \)
- \( \checkmark \) maximum bar diameter,
- \( \checkmark \) 20 mm
Specify **Overall depth of section limit for good anchorage bond** (in mm)

Specify **Minimum distance from top surface for good anchorage** (in mm)

Specify **Minimum anchorage length** (in mm)

Specify **Side cover limit relating to \( \alpha_a \) coefficient** (as diameter)

Specify **Compression flange length factor limit for EC2 Eq. 4.77**

Specify Compression flange breadth factor limit for EC2 Eq. 4.77'

Boxed Values - Shear tab
Specify **Maximum \( \overline{d} \) of links** (max. diameter) (in mm).

Specify **Maximum longitudinal spacing of links when \( VSD \leq \frac{2}{3} VRd2 \)** (in mm).

Specify **Maximum longitudinal spacing of links when \( VSD > \frac{2}{3} VRd2 \)** (in mm).

Specify **Maximum transverse spacing of links** (in mm).

Specify **Upper limit of \( \beta \) used for shear enhancement**

Clause 44.2.3.2.1

Specify \( \sigma_{cp} \)

Specify ’Maximum value of \( \rho_1 \).

**Related Links**

- *D. EHE Beam Design Principles* (on page 1076)
- *D. EHE Column Design Principles* (on page 1077)

D. EHE Column Design Brief dialog
The EHE Column Design Brief has the following tabs for setting out the design parameters:

General tab
Concrete

Specify the concrete strength either from the available concrete grades or by defining a concrete strength, fck, (in N/mm²).

Available concrete grades include: C25, C30, C35, C40, C45 and C50

Aggregate Size

Specify the maximum aggregate size.

Cover

Specify the cover to the shear reinforcement.

Main Bars

Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Available steel grades include: B220S, B400S and B500S.

Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Links

Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.

Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.
**Slenderness Factors**  This may be set to 1, or calculated by the design.  Select value used in clause 43.5.2 from either 0.003 or 0.004'.

**Effective Length Factors**  Specify the effective length factors to be used in the local y-y and z-z directions.

**Starter bars**  Specify the area of starter bar steel (in mm$^2$) to ensure that maximum area of steel is not exceeded by the design.

**Spacing values**  Specify **Maximum allowable spacing between main bars** (in mm).  Specify **Minimum allowable spacing between link bars** (in mm).

---

**Boxed Values - Main tab**
Partial safety factors

Specify the partial safety factors for concrete, gamma(c) and steel, gamma(s).

Sustained compression factor

Specify Sustained compression factor for rectangular and circular columns.

Reinforcement Percentages

Specify Percentage of reinforcement limits for both Minimum and Maximum values.

Minimum no. of bars in a circular section

Specify Minimum number of bars to be used in a circular column.

Minimum transverse dimension of section

Specify Minimum transverse dimension of section (in mm).

Upper slenderness limit for model column method

Specify 'Upper slenderness limit for model column method.

Bar spacing options

Select options for limiting spacing of bars to include:

- Maximum bar diameter
- Specified dimension (in mm).

Boxed Values - Shear tab

Min. diameter of links

Specify Minimum diameter of link bars (in mm), and limiting fraction of diameter of longitudinal bars.

Link spacing options

Select option for minimum spacing of links do not exceed lesser of:

- Multiple of longitudinal bar diameters, specify the number of diameters.
- Least dimension of the column.
- Maximum distance, specify the distance (in mm).

Clause 44.2.3.2.1

Specify the values used in calculating the constant k, the average stress in the concrete due to axial force,
Specify the value of \( \sigma_{cp} \).
Specify the value \( \rho(1) \), maximum value of the reinforcement ratio corresponding to the area of reinforcement effective at the section under consideration.

Member Loadcases tab
Used to select load case results which are used for column design with this brief.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

Note: By default, no load cases or combinations are selected.

Available list  Contains all primary load cases and load combinations in the STAAD.Pro input file.
Selected list  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

Related Links
- D. EHE Beam Design Principles (on page 1076)
- D. EHE Column Design Principles (on page 1077)
D. Using GB50010 (Chinese) Concrete Member Design

Concrete Design

GB50010

Release 2007

Bentley Systems, Inc.
1801 Jiachuang Building, No.2 Qixian Road High-Tech Zone
Dalian, China
Telephone: (0411) 8479-1166
Fax: (0411) 8479-7700
Concrete Design 混凝土构形手册

D. GB50010 Beam Design Brief dialog
The GB50010 Beam Design Brief dialog has the following tabs for setting out the design parameters:

Frame Information tab
Used to provide seismic lateral load resisting frame information for the portion of the structure containing the beam(s) associated with this brief.

Frame Type
Select the type of lateral load resisting system in which the beam(s) associated with this brief are used.
Beam Type

Beam element types, different types of reinforcement will affect the construction requirements.

Earthquake resistant Grade

Select the Grade classification for seismic design.

Select None if the component no seismic testing provisions are used for design (i.e., frame is not considered for seismic resistance).

Earthquake Intensity

Select the intensity level of the design earthquake.

Select None if no Earthquake is considered (i.e., frame is not considered for seismic resistance).

General tab

Minimum Cover
Set the cover to the top, bottom, and sides of the beams (in mm).

Concrete
Select a standard grade of concrete or select the User fcu option to specify a different strength value (in MPa).

Aggregate Size
The maximum aggregate size (in mm) used in the concrete mix.

Envelope
The beams will be designed to the forces from the envelope selected from the pull-down list.

Beam division
Set the minimum number of segments that each member should be divided into for calculating the reinforcement (must be at least four).

Main Reinforcement tab

Used to provided parameters for the longitudinal reinforcing bars.
### Top bar settings
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The Min gap defines the spacing (in mm) between the top 2 layers.

### Bottom bar settings
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links. The Min gap defines the spacing (in mm) between the bottom 2 layers.

### Side Bar Criteria
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

### Main Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the User fy field and set the bending dimensions to be used with that strength.

For User fy values, click the Bending Dimensions button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

---

**Shear Reinforcement tab**

Used to provide parameters for shear reinforcement (e.g., transverse stirrups).
Design Shear Reinforcement

Select the option to design the shear reinforcement up to the center line of the column (support) or column face (edge of support). Note that the column or support face is defined in the Spans Table.

Shear Bar Criteria

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

Select whether the links should be scheduled as Closed or Open.

Shear Bar Type

Either select a standard grade from the pull-down list or enter the steel yield strength into the User fyv (in MPa) and set the bending dimensions to be used with that strength by clicking the Bending Dimensions... button (opens the Bending Dimensions dialog (on page 1157)).

- **OK** Accepts parameters on all dialog tabs and closes the dialog.
- **Cancel** Closes the dialog without saving any changes.
- **Apply** Saves changes made in the dialog. Only available if changes have been made.
- **Help** Opens the STAAD.Pro help window.

Related Links

- [D. Selected Specifications](#) (on page 1025)
- [D. Design Capacity of Beams](#) (on page 1031)
- [D. Design Capacity of Columns](#) (on page 1037)

D. GB50010 Column Design Brief dialog
The GB50010 Column Design Brief dialog has the following tabs for setting out the design parameters:

Frame Information tab

Used to provide seismic lateral load resisting frame information for the portion of the structure containing the column(s) associated with this brief.

**Frame Type**
Select the type of lateral load resisting system in which the column(s) associated with this brief are used.

**Column Type**
Column element types, different types of reinforcement will affect the construction requirements.

- **Frame Col** General columns in a regular frame
- **Transfer Col** General columns in a transfer frame
- **Bot Frm** Columns at the base of a regular frame
- **Bot Trans** Columns at the base of a transfer frame
- **Top Trans** Columns at the top of a transfer frame
- **Corner** Corner Columns of a regular frame
- **Side** Side Columns of a regular frame
- **Middle** Internal Columns of a regular frame

**Earthquake resistant Grade**
Select the Grade classification for seismic design.
Design
D. Concrete Design

Select None if the component no seismic testing provisions are used for design (i.e., frame is not considered for seismic resistance).

**Earthquake Intensity**
Select the intensity level of the design earthquake.
Select None if no Earthquake is considered (i.e., frame is not considered for seismic resistance).

**General Settings tab**
Used to provide parameters for concrete strength, main reinforcement, and shear reinforcement in the columns.

![GB50010 Column Design Brief](image)

**Concrete**
the definition of concrete strength. Aggregate size can be ignored.

**Longitudinal bars, stirrups**
define the type and size of reinforcement.

**Concrete**
Select a standard grade of concrete or select the User fcu option to specify a different strength value (in MPa).

**Aggregate Size**
The maximum aggregate size (in mm) used in the concrete mix.

**Cover**
Set the cover used at the column faces (in mm).

**Main Bars**
Either select a standard grade from the pull-down list or enter the steel yield strength into the User fy field and set the bending dimensions to be used with that strength.

For User fy values, click the Bending Dimensions button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.
Set the Minimum and Maximum allowable bar sizes to be used.

**Links**

Either select a standard grade from the pull-down list or enter the steel yield strength into the **User fyv** (in MPa) and set the bending dimensions to be used with that strength by clicking the **Bending Dimensions...** button (opens the [Bending Dimensions dialog](#) on page 1157).

Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used.

Column Factors tab

Specify column conditions for bracing, effective length, and slenderness.

![GB50010 Column Design Brief](image)

- **Bracing Conditions**: Specify whether the columns are braced in the local y-y and/or z-z directions.
- **Effective Length Factors**: Specify the effective length factors to be used in the local y-y and z-z directions.
- **Slenderness Limits**: This may be set to 1, or calculated by the design.
- **Area provided by starter bars**: Specify the area of starter bar steel (in mm$^2$) to ensure that maximum area of steel is not exceeded by the design.

Member Loadcases tab

Used to select load case results which are used for column design with this brief.
**Available list**  Contains all primary load cases and load combinations in the STAAD.Pro input file.

**Selected list**  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

**Related Links**
- *D. Selected Specifications* (on page 1025)
- *D. Design Capacity of Beams* (on page 1031)
- *D. Design Capacity of Columns* (on page 1037)

**D. IS456**

**D. IS456 Beam Design Brief** dialog

The IS456 Beam Design Brief has the following tabs for setting out the design parameters:

- General tab
Minimum cover
Set the cover to the top, bottom and sides of the beams (in mm).

Minimum Reinforcement Spacing.
Specify the Minimum (in mm)

Beam Design option.
Select either 'Vertical Bending and Shear' or 'Lateral Bending and Shear'

Concrete Grade
Set the strength either from the list of available grades, or by defining a cube strength, fck (in N/mm²).

Available concrete grades include: M20, M25, M30, M35 and M40

Envelope
The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

Divide Beam
Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Design for Torsion
Select whether the beam should include the design for torsion.

Check for IS 13920
Select this option to instruct the program to perform additional seismic checks per IS 13920 for the main bars.

Note: Select the Earthquake page (on page 1216) to check for collapse and shear criteria.

Main Reinforcement tab
### Top Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

### Bottom Bar Criteria
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

### Side Bar Criteria
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

### Main Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

The available steel grades include: Fe250, Fe415 and Fe500

Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

### Shear Reinforcement tab

---

![Concrete Design Interface](image-url)
Design
Shear For
Select whether or not to include for the shear forces within the width of the column (or support) by selecting either 'Center Line of Support' or 'Column face' (support).
Select 'Enhanced Shear Effects' to account for changes in shear angle when approaching a column (support) face.
Select 'Include Axial Load Effects' to account for the axial forces in design of longitudinal reinforcement.

Shear Bar Criteria
Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.
Select if the links are to be detailed as Closed or Open.

Shear Bar Type
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength.
The available steel grades include: Fe250, Fe415, and Fe500.
Click the Bending Dimensions... button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40, and 50

Related Links
- D. IS456 Column Design Principles (on page 1059)
- D. IS456 Beam Design Principles (on page 1057)
- D. IS456 General Considerations (on page 1056)

D. IS456 Column Design Brief dialog
The IS456 Column Design Brief has the following tabs for setting out the design parameters:
General tab

Concrete
Specify the concrete strength either from the available concrete grades or by defining a concrete strength, fck, (in N/mm²).
Specify the minimum center to center spacing for reinforcement.
Specify the cover to the shear reinforcement.
Available concrete grades include: M20, M25, M30, M35 and M40.

Main Bars
Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.
Specify the minimum and maximum bar sizes to be used.
Available steel grades include: Fe250, Fe415 and Fe500.
Click the **Bending Dimensions…** button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Links
Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.
Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.
Available steel grades include: Fe250, Fe415 and Fe500.
Click the **Bending Dimensions…** button to open the Bending Dimensions dialog (on page 1157), which is used to specify bending dimensions for each bar size.

Check for IS 13920
Select this option to instruct the program to perform additional seismic checks per IS 13920 for the main bars.
Design
D. Concrete Design

**Note:** Select the [Earthquake page](#) (on page 1216) to check for collapse and shear criteria.

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Column Parameters tab

**Bracing Conditions**
Specify whether the columns are braced in the local y-y and/or z-z directions.

**Effective Length Factors**
Specify the effective length factors to be used in the local y-y and z-z directions.

**Slenderness Limits**
This may be set to 1, or calculated by the design.

**Area provided by starter bars**
Specify the area of starter bar steel (in mm²) to ensure that maximum area of steel is not exceeded by the design.

Member Loadcases tab

Used to select load case results which are used for column design with this brief.
Available list  Contains all primary load cases and load combinations in the STAAD.Pro input file.

Selected list  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

Note: By default, no load cases or combinations are selected.

Related Links
- D. IS456 Column Design Principles (on page 1059)
- D. IS456 Beam Design Principles (on page 1057)
- D. IS456 General Considerations (on page 1056)

D. NS3473

D. NS 3473 Beam Design Brief dialog
The NS 3473 Beam Design Brief has the following tabs for setting out the design parameters:

General tab
Minimum cover  Set the cover to the top, bottom and sides of the beams (in mm).

Aggregate Size  Maximum size of aggregate to be used (in mm)

Concrete Grade  Set the strength either from the list of available grades, or by defining a cube strength, fck (in N/mm²).

   Available concrete grades include: >C25, C35, C45, C55, C65, C75 and C85

Envelope  The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

Divide Beam  Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

Main Reinforcement tab
Criteria

**Top Bar Criteria**  Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

**Bottom Bar Criteria**  Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

**Side Bar Criteria**  Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

**Main Bar Type**  Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

The available steel grades include: B400, B500 and B600

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Shear Reinforcement tab
**Design**

**D. Concrete Design**

- **Design Shear For**
  - Select whether or not to include for the shear forces within the width of the column (or support) by selecting either **center Line of Support** or **Column face** (support).
  - Select **Include Axial Load Effects** to account for the axial forces in design of longitudinal reinforcement.

- **Shear Bar Criteria**
  - Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

- **Shear Bar Type**
  - Either select a standard grade from the pull-down list or enter the steel yield strength into the user value (in N/mm²) and set the bending dimensions to be used with that strength.
  - The available steel grades include: S400 or S500

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Boxed Values - General tab
**Design Factors**  
Specify partial safety factors for Concrete, Gamma(c) and Steel, Gamma(s)  
Specify additional factor for sustained compression, alpha  
Limit for maximum percentage of tension and compression reinforcement (as %)

**Equation 4.78**  
Specify values for k factor, f(ct eff) (in N/mm²) and sigma(s) (as %)

**Bar Spacing**  
Set the minimum spacing of bars (in mm) to be used on top of the 2x bar size limit.

![Boxed Values - Anchorage & Slender](image)

**Boxed Values - Anchorage & Slender tab**

<table>
<thead>
<tr>
<th>Partial safety factors</th>
<th>Concrete $\gamma_c$</th>
<th>Steel $\gamma_s$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.40</td>
<td>1.25</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Additional factor for sustained compression</th>
<th>$\alpha$</th>
<th>0.85</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Maximum percentage of tension and compression reinforcement</th>
<th>%</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4.00</td>
</tr>
</tbody>
</table>

**Equation 4.78**

<table>
<thead>
<tr>
<th>k factor</th>
<th>f(ct eff)</th>
<th>$\sigma_s$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.80</td>
<td>3.0 N/mm²</td>
<td>100 % of $f_{yk}$</td>
</tr>
</tbody>
</table>

Spacing of bars is not less than 2x maximum bar diameter or 20 mm.
<table>
<thead>
<tr>
<th>Specification</th>
<th>Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overall depth of section limit for good anchorage bond</td>
<td>250 mm</td>
</tr>
<tr>
<td>Minimum distance from top surface for good anchorage</td>
<td>300 mm</td>
</tr>
<tr>
<td>Minimum anchorage length</td>
<td>100 mm</td>
</tr>
<tr>
<td>Side cover limit relating to $\alpha_a$ coefficient</td>
<td>3.00</td>
</tr>
<tr>
<td>Compression flange length factor limit for EC2 Eq. 4.77</td>
<td>50.00</td>
</tr>
<tr>
<td>Compression flange breadth factor limit for EC2 Eq. 4.77’</td>
<td>2.50</td>
</tr>
</tbody>
</table>

Specify **Overall depth of section limit for good anchorage bond** (in mm)
Specify **Minimum distance from top surface for good anchorage** (in mm)
Specify **Minimum anchorage length** (in mm)
Specify **Side cover limit relating to alpha($\alpha_a$) coefficient** (as diameter)
Specify **Compression flange length factor limit for EC2 Eq. 4.77**
Specify Compression flange breadth factor limit for EC2 Eq. 4.77’

Boxed Values - Shear tab
Specify **Maximum diameter of links** (in mm)

Specify **Maximum longitudinal spacing of links when VSD <= 2/3 VRd2** (in mm)

Specify **Maximum longitudinal spacing of links when VSD > 2/3 VRd2** (in mm)

Specify **Maximum transverse spacing of links** (in mm)

Specify **Upper limit of beta used for shear enhancement**

Equation 4.18

- Specify $k$ to be used and limit of $k \cdot d$
- Specify $\sigma_{cp}$
- Specify 'Maximum value of $\rho_1$

Related Links
- [D. NS3473 Beam Design Principles](#) (on page 1065)
- [D. NS3473 Column Design Principles](#) (on page 1066)

D. NS 3473 Column Design Brief dialog

The NS 3473 Column Design Brief has the following tabs for setting out the design parameters:

General tab
**Concrete**  Specify the concrete strength either from the available concrete grades or by defining a concrete strength, fck, (in N/mm²).

Specify the maximum aggregate size.

Specify the cover to the shear reinforcement.

Available concrete grades include: C25, C35, C45, C55, C65, C75 and C85

**Main Bars**  Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Available steel grades include: B400, B500 and B600.

**Links**  Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.

Available steel grades include: B400, B500 and B600.

Unless stated otherwise, the range of available bars sizes includes: 12, 16, 20, 25, 32 and 40.

Column Factors tab
Slenderness Factors  This may be set to 1, or calculated by the design.

Specify Creep coefficient

For long term loads, specify Longitudinal Force (in kN) and First order moment (in kNm)

Effective Length Factors  Specify the effective length factors to be used in the local y-y and z-z directions.

Area provided by starter bars  Specify the area of starter bar steel (in mm²) to ensure that maximum area of steel is not exceeded by the design.

Specify Maximum allowable spacing between main bars (in mm).

Specify Minimum allowable spacing between link bars' (in mm).

Boxed Values - Main tab
Specify **Partial safety factors** for Concrete, \( \gamma_c \) and Steel, \( \gamma_s \).

Specify **Sustained compression factor** for rectangular and circular sections.

Specify **Reinforcement percentages** both minimum and maximum values.

Specify **Minimum number of bars to be used in circular columns**

Specify **Minimum transverse dimension of section** (in mm)

Specify **Upper slenderness limit for model column method**

Specify **Minimum spacing of bars** on top of 2x bar limit.

Boxed Values - Shear tab

Specify **Minimum diameter of link** (in mm) and fraction of diameter of longitudinal bars.

### Design

D. Concrete Design

-----

**Boxed Values - Shear**

**Column Factors**

**Boxed Values - Main**

---

<table>
<thead>
<tr>
<th>Partial safety factors</th>
<th>Concrete ( \gamma_c )</th>
<th>Steel ( \gamma_s )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.40</td>
<td>1.25</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Sustained compression factor</th>
<th>Rectangular</th>
<th>Circular</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.85</td>
<td>0.85</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Reinforcement percentages</th>
<th>Min value of</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>min</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.00</td>
<td>8.00</td>
</tr>
</tbody>
</table>

| Minimum no. of bars in a circular section | 6        |
| Minimum transverse dimension of section  | 200 mm    |

Upper slenderness limit for model column method 140.00
Spacing of bars is not less than 2x maximum bar diameter or 20 mm

<table>
<thead>
<tr>
<th>Min diameter of links is</th>
<th>6 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>or 0.25 ( x ) max diameter of longitudinal bars</td>
<td></td>
</tr>
</tbody>
</table>

Spacing of links does not exceed the lesser of:
- multiple of longitudinal bar diameter 12
- the least dimension of the column
- minimum distance 300 mm

Equation 4.18

\[
k = 1.50 - d + \frac{0.00}{\phi} - \frac{0.02}{1}
\]
Select whether options to limit spacing of links as:

- a user number diameters of longitudinal bar diameters.
- the least dimension of the column.
- a user dimension.

**Equation 4.18** Specify a value to use in k factor equation, with a lower limit

Specify \( \sigma_{cp} \)

Specify **Maximum value of \( \rho(1) \)**

**Member Loadcases tab**

Used to select load case results which are used for column design with this brief.

**Available list** Contains all primary load cases and load combinations in the STAAD.Pro input file.

**Selected list** Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the <strong>Selected</strong> list.</td>
</tr>
</tbody>
</table>
Note: By default, no load cases or combinations are selected.

Related Links
- D. NS3473 Beam Design Principles (on page 1065)
- D. NS3473 Column Design Principles (on page 1066)

D. SP 52-101-03 Briefs
design per Russian Concrete code: SP 52 101-03 "Concrete and reinforced concrete structures without prestressing" (СП 52-101-2003 "Бетонные и железобетонные конструкции без предварительного напряжения арматуры").

Note: The Russian Design Codes are only available with the ‘Euro-zone – Eastern Design Codes’ SELECT license.

General Tab

Concrete grade
Select a standard grade from the pull-down list.

Minimum Cover
Specify the cover to the top, bottom and sides of the beams (in mm).

Partial Safety Factors
Specify safety factors to be used in the design for both concrete and steel.

Design for (shear at)
Select whether or not to include for the shear forces within the width of the column (or support) by selecting either Center Line of Support or Column Face (support).
**Design for Torsion**  
Select this option to consider torsion in the design of the reinforcement.

**Envelope**  
The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the [Define Envelope dialog](#) (on page 1205).

**Divide Beam into**  
Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

**Reinforcement Tab**

![Reinforcement Tab](image)

**Main Bar Criteria**  
Select the Steel grade

Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams in the **Min Size** and **Max Size** fields, respectively. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The **Minimum gap between bars** defines the spacing (in mm) between the top 2 layers.

**Shear Bar Criteria**  
Select a standard grade from the pull-down list.

Select the minimum bar **Size** and **No. of Legs** to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

**Sag Bar Diameters**  
Select the criteria for selecting the sag bar diameters from either **Equal to hog bars**, **Optimized for each span independently**, or **Optimized for sag-critical span**.

**OK**  
Accepts parameters on all dialog tabs and closes the dialog.

**Cancel**  
Closes the dialog without saving any changes.

**Apply**  
Saves changes made in the dialog. Only available if changes have been made.

**Help**  
Opens the STAAD.Pro help window.

**Related Links**
- *D. SP52-101-03 Beam Design Principles* (on page 1067)
D. SP 52 101-03 Column Design Principles

D. SP 52 101-03 Column Brief dialog
Design per Russian Concrete code: SP 52 101-03 "Concrete and reinforced concrete structures without prestressing" (СП 52-101-2003 "Бетонные и железобетонные конструкции без предварительного напряжения арматуры").

**Note:** The Russian Design Codes are only available with the 'Euro-zone – Eastern Design Codes' SELECT license.

General tab

- **Concrete**
  - Select a standard grade from the pull-down list. Specify the cover to the top, bottom and sides of the beams (in mm).

- **Effective Length Factors**
  - Specify the effective length factors to be used in the local y-y and z-z directions.

- **Partial Safety Factors**
  - Specify safety factors to be used in the design for both concrete and steel.

- **Equations for Circular Columns**
  - Select the methodology to be used for calculating the capacity of circular columns: Non-linear theory, SNiP 1984, or SNiP Alternative.
Main Bar Criteria

Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used (in MPa). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum and maximum bar sizes to be used.

Shear Bar Criteria

Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used (in MPa). If a user strength is defined, then the appropriate bar bending dimensions need to be specified.

Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.

Note: Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

Member Loadcases tab

Used to select load case results which are used for column design with this brief.
Available list  Contains all primary load cases and load combinations in the STAAD.Pro input file.

Selected list  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the Available list to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the Available list to the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the Selected list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

**OK**  Accepts parameters on all dialog tabs and closes the dialog.

**Cancel**  Closes the dialog without saving any changes.

**Apply**  Saves changes made in the dialog. Only available if changes have been made.

**Help**  Opens the STAAD.Pro help window.

**Related Links**
- *D. SP52-101-03 Beam Design Principles* (on page 1067)
- *D. SP52-101-03 Column Design Principles* (on page 1070)

*D. TS500*
D. TS500 Beam Design Brief dialog

The TS500 Beam Design Brief has the following tabs for setting out the design parameters:

**General tab**

- **Minimum cover**
  - Set the cover to the top, bottom and sides of the beams (in mm).

- **Concrete**
  - Set the strength either from the list of available grades, or by defining a cube strength, fck (in N/mm²).
  
  Available concrete grades include: C20, C25, C30, C35, C40, C45 and C50

  Specify the maximum size of aggregate to be used (in mm)

- **Forces**
  - Select whether to design the forces within the width of the column (support) or only up-to the face of the column (support).

- **Envelope**
  - The beams will be designed to the forces from the envelope selected from the pull-down list. Load Envelopes are added using the Define Envelope dialog (on page 1205).

- **Divide Beam**
  - Set the minimum number of segments that each member should be divided into for calculating the reinforcement.

**Main Reinforcement tab**
**Top Bar Criteria**  
Set the Minimum and Maximum allowable bar sizes to be used on the top of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the top 2 layers.

Available bar sizes include: 12, 16, 20, 25, 32, 40 and 50

**Bottom Bar Criteria**  
Set the Minimum and Maximum allowable bar sizes to be used on the bottom of the beams. The link hanger size is used for detailing where no reinforcement bars are required for bending, but bars are required for supporting links.

The Min gap defines the spacing (in mm) between the bottom 2 layers.

Available bar sizes include: 12, 16, 20, 25, 32, 40 and 50

**Main Bar Type**  
Either select a standard grade from the pull-down list or enter the steel yield strength into the user value and set the bending dimensions to be used with that strength.

The available steel grades include: S220a, S420a or S500a

If the **User fyk** option is selected, specify a size. The bend radii can be specified in the Bending Dimensions dialog (on page 1157) by clicking the Bending Dimensions... button.

Select whether the ‘Development Length’ is taken from the ACI 318 code, clause 12.2.2 or 12.2.3

Select the **Epoxy Coated** option if the development length calculations should consider this reduction.

**Side Bar Criteria**  
Set either the minimum bar size to be used or let the program calculate which bars can be used to achieve a suitable spacing on deep beams.

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50
Shear Reinforcement tab

**Design Shear Force**
Select whether or not to include for the shear forces within the width of the column (or support) by selecting either 'center Line of Support' or 'Column face' (support).

Select 'Include Axial Load Effects' to account for the axial forces in design of longitudinal reinforcement.

**Shear Bar Criteria**
Select the minimum bar size and number of legs (2, 3, 4, 5, or 6) to be used and set the minimum spacing (in mm). If the design requires a smaller spacing, then the number of legs will be increased.

**Shear Bar Type**
Select either a standard grade from the list or **User fyks** to specify the steel yield strength (in N/mm²). If the User fyks option is selected, you may also specify custom minimum bending radii to be used with that strength in the Bending Dimensions dialog (on page 1157) by clicking the Bending Dimensions... button.

The available steel grades include: S220a, S420a and S500a

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

**OK**
Accepts parameters on all dialog tabs and closes the dialog.

**Cancel**
Closes the dialog without saving any changes.

**Apply**
Saves changes made in the dialog. Only available if changes have been made.

**Help**
Opens the STAAD.Pro help window.

**Related Links**
- **D. TS 500 Beam Design Principles** (on page 1078)
- **D. TS 500 Column Design Principles** (on page 1080)
D. TS500 Column Design Brief dialog
The TS-500 Column Design Brief has the following tabs for setting out the design parameters:

General tab

Concrete
Specify the concrete strength either from the available concrete grades or by defining a concrete strength, fck, (in \( N/mm^2 \)).
Specify the maximum aggregate size, (in mm)
Specify the cover to the shear reinforcement, (in mm)
Available concrete grades include: C20, C25, C30, C35, C40, C45 and C50

Main Bars
Specify the strength of steel to be used for the main bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.
Specify the minimum and maximum bar sizes to be used.
Available steel grades include: S220a, S420a and S500a.

Links
Specify the strength of steel to be used for the shear reinforcement bars either from the available steel grades or by specifying the steel yield strength to be used. If a user strength is defined, then the appropriate bar bending dimensions need to be specified.
Specify the minimum bar size to be used from 6, 8, 12, 16 or 20.
Select whether for circular columns either 'Tie' or 'Spiral' reinforcement should be used.
Available steel grades include: S220a, S420a and S500a
**Include Seismic Checks**
Select this option to include additional checks.

**Note:** Unless stated otherwise, the available bar sizes include: 6, 8, 10, 12, 16, 20, 25, 32, 40 and 50

**Column Factors tab**

- **Biaxial β factor**
  - Specify the value for the biaxial beta factor.

- **R_m**
  - Specify the value for the R(m) factor.

- **Include Torsion Effects**
  - Select this option to include torsion effects.

- **Design as Sway Frame**
  - Select this option to Design as Sway Frame.

- **Starter Bar Area Provided**
  - Specify the area of starter bar steel (in mm²) to ensure that maximum area of steel is not exceeded by the design.

**Slenderness Factors**
This may be set to 1, or calculated by the design.

Specify 'Creep coefficient'
For long term loads, specify 'Longitudinal Force' (in kN) and 'First order moment' (in kNm)

**Bracing Conditions**
in both local y-y and z-z directions.

Specify the effective length factors to be used.
Select whether or not the local axis is braced, and if not, the value of stability Index Q to use.
Select the load case to be used for Sidesway or use the same as the design load case.
Used to select load case results which are used for column design with this brief.

**Available list**  Contains all primary load cases and load combinations in the STAAD.Pro input file.

**Selected list**  Load cases and combinations the results of which will be used for the column design when this brief is used in a Design Group.

<table>
<thead>
<tr>
<th>Tool icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Adds the load case(s) selected in the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Adds all load cases from the <strong>Available</strong> list to the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Removes the selected load case(s) from the <strong>Selected</strong> list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Removes all load case(s) from the <strong>Selected</strong> list.</td>
</tr>
</tbody>
</table>

**Note:** By default, no load cases or combinations are selected.

**Related Links**
- *D. TS 500 Beam Design Principles* (on page 1078)
- *D. TS 500 Column Design Principles* (on page 1080)

**D. Advanced Concrete Design**

This workflow launches the STAAD.Pro Advanced Concrete Design (RCDC) program, which is used for the design and detailing of reinforced concrete building structures.
The Advanced Concrete Design workflow provides direct access for STAAD.Pro models to leverage the power of the RCDC application. This is a standalone application, which is operated outside the STAAD.Pro environment, but requires a model and results data from a suitable analysis.

The model should typically be formed from beams and columns (plates are currently not supported). RCDC can be used to design the following objects:

- Pile Caps
- Footings
- Columns and walls (note walls are determined from a STAAD model as very wide columns, see the RCDC documentation for more details)
- Beams
- Slabs (zones defined by a loop of beams, not plates, see RCDC documentation for more details)

As projects progress, each design created in RCDC is retained and displayed when RCDC is reentered so that previous designs can be recalled and/or continued. If any analysis is performed on the STAAD.Pro model from when the RCDC document was last edited, this is identified as an old document and should be reviewed again to ensure that the results of the design are still valid.

**Note:** When using the model in STAAD.Pro Advanced, access to RCDC will be provided using the same STAAD.Pro Advanced license and no other license is required. If using STAAD.Pro (basic), then the Advanced Concrete Design workflow will still allow access to RCDC, but this will require the use of an additional license. This could either be a STAAD.Pro Advanced license or a STAAD Advanced Concrete license. Please check which licenses are available to you before attempting to use this and ensure that you use the correct license.

**D. Advanced Slab Design**

The STAAD.Pro Advanced Slab Design workflow is an integrated tool that works from within the STAAD.Pro environment. Concrete slabs can be defined and the data can be transferred to RAM Concept.

The data passed into RAM Concept includes the geometry, section and material properties, loads and combination information, and analysis results.

**Note:** This feature requires STAAD.Pro 2006 Build 1001 or higher and is compatible with RAM Concept version v2.1 and higher.

Basic reinforced concrete slab design per BS8110, ACI 318-05/ 318M-05, or EC2:2004 can also be performed using the **D. Interactive Concrete Design** (on page 1005).

**Note:** Beginning with STAAD.Pro CONNECT Edition, direct export to ADAPT-Builder® has been deprecated.

**D. Using the Advanced Slab Design workflow**

**Note:** A successful analysis of a STAAD.Pro input file is required before initiating and using the Advanced Slab Design workflow.

**D. To open the Advanced Slab Design workflow**

1. Open an input file (containing the slab to be designed) in STAAD.Pro.
2. Perform a successful analysis.
3. Select Advanced Slab Design in the Workflows panel.

D. To create a load envelope

1. Select the Envelopes page in the Advanced Slab Design page control bar.
The Envelopes table.

   **Tip:** This page is typically open by default.

2. Click New Envelope on the Envelopes table.
   workflow.
The Load Envelope dialog opens.

3. (Optional) Type a title for the load envelope.
   A Load Envelope name should be entered, being defined by the chosen load cases by clicking the check boxes corresponding to each load case.

4. Select the load cases or combinations to be included in the design load envelope.

   **Tip:** Check the Select All Load Cases Shown Below option to add all available load cases to the envelope.

5. For each load case included, select a Load Type to describe the nature of that load.
6. Click OK.
The new load envelope is added to the Envelopes table.

Related Links
- D. Envelopes table (on page 1356)
- D. Load Envelope dialog (on page 1356)

D. To create a slab definition

1. Select the Slab Design page in the Advanced Slab Design page control bar.
The Geometry Cursor tool is selected.

2. Select the structure geometry which will form the slab.

   **Note:** The slab geometry should include all plate elements which form the slab, as well as column members both above and below the slab, beam members framing the slab, and wall plate elements which are connected both above and below to the slab.

3. Click New Slab on the Slabs table.
The Slab Definition dialog opens.

4. Select the Load Envelope with which to associate the slab definition.
5. Click OK.
The new slab definition is added to the Slabs table.

   **Note:** If the selection of plates is such that the plates form two separated entities (e.g., plates that form parallel floors) then STAAD.Pro will create multiple separate slab entities.
D. To export slab definitions to RAM Concept

1. Select the slab design definition(s) in the Slabs table.
2. Select the Export > Export RAM Concept Information tool in the RAM Concept group on the Advanced Slab Design ribbon tab.

A dialog opens to confirm that the data files were created.

D. To open the STAAD.Pro slab data in RAM Concept

1. Select the slab design definition(s) in the Slabs table.
2. Either:
   Select the Export > Run RAM Concept tool in the RAM Concept group on the Advanced Slab Design ribbon tab.

   or

   Right-click on the Slabs table and select Run > RAM Concept from the pop-up menu.
   RAM Concept launches and The Concept import dialog opens.
3. Leave all structure geometry element types selected (default) and click OK.
   Upon the completion of a successful import, a message dialog opens to inform you of the total number of structural elements and loads created.

Refer to the RAM Concept documentation for assistance performing analysis and design operations.

D. Pages in the Advanced Slab Design workflow

The Pages in the Advanced Slab Design workflow are described below in brief.

Table 85: Page Controls in the Advanced Slab Design workflow

<table>
<thead>
<tr>
<th>Page</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Envelopes</td>
<td>Used to create set(s) from available Primary and Combination Loads defined in the current STAAD.Pro file for which analysis results are already available. When the Envelopes page is selected, the Envelopes table opens.</td>
</tr>
</tbody>
</table>
### Slab Design

<table>
<thead>
<tr>
<th>Page</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slab Design</td>
<td>Used to create Slab definitions for slab design. When the <strong>Slab Design</strong> page is selected, the <strong>Slabs</strong> table opens.</td>
</tr>
</tbody>
</table>

**D. Envelopes** table

Used to view a table containing all the load envelopes created by the user for the current model. This table will also open even if there is no load envelope created yet.

Opens when the **Envelopes** page is selected in the **Advanced Slab Design** workflow.

**Envelopes list**  Contains a list of all Envelopes available for Slab Design, along with the component load cases.

**New Envelope**  Opens the **Load Envelope** dialog (on page 1356), which is used to create a load envelope from defined loads.

**Edit Envelope**  Opens the **Load Envelope** dialog for editing the selected envelope.

**Related Links**

- **D. To create a load envelope** (on page 1354)

**D. Load Envelope** dialog

Used to define an envelope of primary and combined load cases used in the analysis that will then be used by RAM Concept for performing code checks or member selections.

Opens when **New Envelope** is clicked on the **Envelopes table** (on page 1356).
Envelope Name

Specify a title for the load envelope. The default title uses an incremented id number.

Select All Load Cases Shown Below

Select this option to add all load cases in the Defined Loads list to the new load envelope.

Defined Loads list

Lists all previously defined load cases in the input file. For each load selected for the Load Envelope, the Load Type must be defined.

The Load Type describes the nature of the load. Clicking in the cell for a Load Case opens a drop-down list containing all load types. Select the most appropriate type to make the assignment.

OK

Creates a new Load Envelope with the selected parameters and closes the dialog.

Cancel

Closes the dialog without creating a new load envelope.

Related Links

- D. To create a load envelope (on page 1354)

D. Slabs table

Contains the slabs already defined along with the associated design load envelope.

Opens when the Slab Design page is selected in the Advanced Slab Design workflow.
Slab no Lists sequential slab reference numbers (S1, S2, etc.)
Slab Name Name of the Slab definition.
Envelope name The load envelope associated with the slab.

New Slab Opens the Slab Definition dialog (on page 1358), which is used to create a slab definition from a set of selected plate elements.

Edit Slab Opens the Slab Definition dialog to edit or rename the currently selected slab.

Related Links
• D. To create a slab definition (on page 1354)

D. Slab Definition dialog

Used to create a slab definition from a set of selected geometry.

Opens when New Slab is clicked on the Slabs table (on page 1357).

Slab name (Edit mode only) Enter the name of the Slab definition. Slab definitions are automatically named with incrementing numbers when they are created.
Load Envelope Select a previously defined design load envelope from the list.
OK Closes the dialog and creates a slab from the selected geometry.
Cancel Closes the dialog without creating a slab definition.

Related Links
• D. To create a slab definition (on page 1354)
D. Foundation Design

The Foundation Design workflow integrates STAAD Foundation Advanced into STAAD.Pro. In this workflow, you can select loads and geometry to pass to a new STAAD Foundation Advanced project.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 2) (release 20.07.07) or higher.

**Tip:** Models containing a large number of supported nodes or load cases may result in slow performance on older computer hardware. Exporting a limited set of data can be used to improve performance in STAAD Foundation Advanced in these cases.

D. Using the Foundation Design workflow

**Note:** See Limited Versus Full Licensed Versions of STAAD Foundation Advanced (on page 1359) for details on the two license options available and the foundations that can be designed using those licenses.

Limited Versus Full Licensed Versions of STAAD Foundation Advanced

STAAD Foundation Advanced is available in two forms:

- licensed version
- a limited version which may be used with a STAAD Foundation Advanced license

**Note:** This limited version requires a license for STAAD.Pro but not a separate license for STAAD Foundation Advanced.

When you launch the program with STAAD Foundation Advanced installed, you are given a choices to select one of these in the **License Configuration** dialog. If you select the limited version option, then you only have access
to the items as indicated in the following table. If you have purchased a license then you will have access to all the modules in STAAD.Pro.

**Note:** All available design codes are included for each feature.

**Table 86: Items included in the free, limited version of STAAD Foundation Advanced**

Items indicated with a “✔” are available with a free, limited version license. Items indicated with an “X” require a license to use.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Feature Name</th>
<th>Limited Version (license free)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Toolkit</td>
<td>All modules</td>
<td>X</td>
</tr>
<tr>
<td>General</td>
<td>Isolated footings</td>
<td>✔</td>
</tr>
<tr>
<td>General</td>
<td>Combined footings</td>
<td>✔</td>
</tr>
<tr>
<td>General</td>
<td>Pilecaps</td>
<td>✔</td>
</tr>
<tr>
<td>General</td>
<td>Mat foundations</td>
<td>X</td>
</tr>
<tr>
<td>General</td>
<td>Octagonal footings</td>
<td>X</td>
</tr>
<tr>
<td>General</td>
<td>Strap footings</td>
<td>X</td>
</tr>
<tr>
<td>General</td>
<td>Vibrating machine foundations</td>
<td>X</td>
</tr>
<tr>
<td>General</td>
<td>Combined with rib</td>
<td>X</td>
</tr>
<tr>
<td>Plant</td>
<td>All modules</td>
<td>X</td>
</tr>
</tbody>
</table>

**Tip:** If you uncheck the **Show on startup** option but need to access the **License Configuration** dialog, you can select the **License Configuration** tool in the **Settings** group on the **Home** ribbon tab.

D. To open the Foundation Design workflow

1. Perform a successful analysis.
2. Select **Foundation Design** in the **Workflows** panel.

The **Foundation Design** dialog opens.

D. To export all of the structure data to a STAAD Foundation Advanced project

This is the typical method of transferring data. Keep in mind that you can set up multiple jobs in STAAD Foundation Advanced, each containing some or all of the STAAD.Pro supports or load cases.
1. In the **Foundation Design Options** dialog, select the **All Supports** option if it is not already selected.
2. Click **Include All** to transfer all available load cases to the list of load cases to be included in design.
3. Click **STAAD Foundation Advanced**.
   STAAD Foundation Advanced opens and the STAAD.Pro support data and results are imported.

**D. To export a limited set of structure data to a STAAD Foundation Advanced project**

1. Either:
   - Use the **Node Cursor** tool (selected by default with then Foundation Design workflow is selected) to graphically select the support nodes you want to transfer. The **Selected Supports** option will be selected automatically in the **Foundation Design** dialog.
   - or
   - Select the **Listed Supports** option in the **Foundation Design** dialog and then type a list of node numbers.
2. Either:
   - Select one or more load cases in the **Excluded from design** list and click **Include** to transfer them to the **To be included in design** list.
   - or
   - Select the **Select Envelope** option and then select a load envelope from the drop-down list.
3. Click **STAAD Foundation Advanced**.
   STAAD Foundation Advanced opens and the STAAD.Pro support data and results are imported.

**Tip:** To change the loads or update the STAAD Foundation Advanced model per any geometry changes to your STAAD.Pro model, simply re-perform the steps involved in exporting the data.

**D. Foundation Design** dialog

Used to specify the supports and results to be exported to a STAAD Foundation Advanced project.

Opens when the **Foundation Design** workflow is selected.

<table>
<thead>
<tr>
<th><strong>Supports for Foundation Design</strong></th>
<th><strong>Load Cases and Combinations</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>• <strong>All Supports</strong> – All supports in the structure will be exported to the STAAD Foundation Advanced project.</td>
<td>Select the load cases to be included in the export to STAAD Foundation Advanced project:</td>
</tr>
<tr>
<td>• <strong>Selected Supports</strong> – Only supports which are selected in the Active view window will be included. Use the <strong>Nodes Cursor</strong> tool to make selections.</td>
<td>• <strong>Select Envelope</strong> – select a load envelope from the drop-down list.</td>
</tr>
<tr>
<td>• <strong>Listed Supports</strong> – List the support numbers of supports to be included.</td>
<td><strong>Tip:</strong> You can create envelopes in the Analytical Modeling workflow. Refer to <strong>M. To create a load envelope</strong> (on page 877) for details.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Excluded from design</strong></th>
<th><strong>Select Load Cases and Combinations</strong> – select this option to choose load cases and combinations to include in the design.</th>
</tr>
</thead>
<tbody>
<tr>
<td>All primary load cases and load combinations are listed here. Any that remain in this list are not exported to the STAAD Foundation Advanced project.</td>
<td></td>
</tr>
</tbody>
</table>
Table 87: Load Selector tools

<table>
<thead>
<tr>
<th>Click this button</th>
<th>To</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Include</strong></td>
<td>Include load cases selected in the Excluded from design list in the To be included in design list below.</td>
</tr>
<tr>
<td><strong>Include All</strong></td>
<td>Include all available load cases in the To be included in design list below.</td>
</tr>
<tr>
<td><strong>Exclude</strong></td>
<td>Remove the selected load cases from the To be included in design list below.</td>
</tr>
<tr>
<td><strong>Exclude All</strong></td>
<td>Remove all load cases in the To be included in design list below.</td>
</tr>
</tbody>
</table>

To be included in design

Loads included here will be exported to the STAAD Foundation Advanced project.

STAAD Foundation Advanced

Starts STAAD Foundation Advanced and imports the structural data selected in this dialog into a new project file.

D. Aluminium Design

D. Available Aluminum Design Codes

The following aluminum design code is available in batch mode in STAAD.Pro.

Table 88: Aluminum Design codes available in STAAD.Pro

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>US</td>
<td><strong>ASD 1994</strong> (on page 1543)</td>
<td>Specifications for Aluminum structures</td>
</tr>
</tbody>
</table>

Related Links

- *D. To specify aluminum design code and parameters* (on page 1363)
- *D. Batch Design versus Interactive Design Workflows* (on page 944)
D. To specify aluminum design code and parameters

To initiate the design of aluminum members and specify the code parameters, use the following procedure.

**Note:** To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

1. On the Analysis and Design ribbon tab, select Aluminium in the Design gallery.
   The Aluminum Design - Whole Structure dialog opens.
   **Note:** Only the US Aluminum code is available for design of aluminum members.
2. Click Define Parameters.
   The Design Parameters dialog opens.
3. Specify a value or option for each required parameter for a set of members and then click Add.
   You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.
   **Note:** Different parameters can be used for different member type designs (e.g., columns, beams, etc.). Alternatively, you can use a separate set of parameters for different member types.
4. Click Close.
   The design parameters are marked with an icon. This indicates that the need to be assigned to members.
5. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.

**Related Links**
- *D. Available Aluminum Design Codes* (on page 1362)
- *Aluminum Design - Whole Structure dialog* (on page 3086)
- *Design Parameters dialog* (on page 3083)
- *TR.48.1 Parameter Specifications* (on page 2851)

D. To specify aluminum design commands

To specify the code checking or design commands to be used for the aluminum design operation, use the following procedure.

1. On the Aluminum Design - Whole Structure dialog, click Commands.
   The Design Commands dialog opens.
2. To perform a design action, select one of the following options (as they are available for the code selection):
   **Select...**
   - check the capacity of the current member sizes
     the CHECK CODE tab
   - select a member size based on the last analysis results
     the SELECT tab
3. Click Add.
4. Click Close.
5. Assign the design commands to the appropriate members.
D. To generate aluminum take off

To generate a summary of all aluminum sections with their lengths and weights for the whole structure or a list of members, use the following procedure.

2. Select the TAKE OFF tab. This will generate a list of sections used including total length and weight.
3. Click Add.
4. Click Close.
5. Assign the take off command to members or to a named group of members.

D. Timber Design

D. Available Timber Design Codes

The following timber design codes are available in batch design in STAAD.Pro.

Table 89: Timber Design codes available in STAAD.Pro

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Canada</td>
<td>CSA 086-01</td>
<td>Wood Design Standard</td>
</tr>
</tbody>
</table>
D. To specify timber design code and parameters

To initiate the design of timber members and specify the code parameters, use the following procedure.

**Note:** To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

1. On the Analysis and Design ribbon tab, select Timber in the Design group gallery.
   The Timber Design - Whole Structure dialog opens.
2. In the Timber Design - Whole Structure dialog, select the applicable concrete design code from the Current Code drop-down list.
3. Click Define Parameters.
   The Design Parameters dialog opens.
4. Specify a value or option for each required parameter for a set of members and then click Add.
   You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.

**Note:** Different parameters can be used for different member type designs (e.g., columns, beams, etc.). Alternatively, you can use a separate set of parameters for different member types.

5. Click Close.
   The design parameters are marked with an icon. This indicates that the need to be assigned to members.
6. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.

**Related Links**
- TR.52 Timber Design Specifications (on page 2857)
- D. Available Timber Design Codes (on page 1364)
- Timber Design - Whole Structure dialog (on page 3088)
- Design Parameters dialog (on page 3083)

D. To specify timber design commands

To specify the code checking or design commands to be used for the timber design operation, use the following procedure.

   The Design Commands dialog opens. The tabs (if any) available in this dialog are dependant on the current design code selection. Not all commands are available for every design code.
2. To perform a design action, select one of the following options (as they are available for the code selection):
To...

check the capacity of the current member sizes

select a member size based on the last analysis results

Select...

the CHECK CODE tab

the SELECT tab (AITC 1984 code only)

3. Click Add.
4. Click Close.
5. Assign the design commands to the appropriate members.

Related Links
- Timber Design - Whole Structure dialog (on page 3088)
- Design Commands dialog (on page 3084)

D. Design Codes

This section contains engineering reference material and input commands for structural design using batch mode. The codes are organized by country, material, and edition.

D1. American Codes

D1.A. American Codes - Steel Design per AISC 360 Unified Specification

Steel member design per ANSI/AISC 360-05, 360-10, and 360-16 Specifications for Structural Steel Buildings, is available in STAAD.Pro. These specifications are published as part of the AISC Steel Construction Manual. Since the ASD and the LRFD method are both addressed in those specifications, they are referred to as UNIFIED.

To use the 2016 edition (default), specify the command:

CODE AISC UNIFIED

or

CODE AISC UNIFIED 2016

To use the 2010 edition, specify the command:

CODE AISC UNIFIED 2010

To use the 2005 edition, specify the command:

CODE AISC UNIFIED 2005

Tip: Either method may be selected in the user interface using the Steel Design - Whole Structure dialog.

Design can be performed according to the provisions for Load and Resistance Factor Design (LRFD) or to the provisions for Allowable Strength Design (ASD), as per section B3 of the code. This selection of the design methodology can be done through the METHOD parameter. The full list of parameters is given in D1.A.6 Design Parameters (on page 1378).

Related Links
- G.17.2.1.4 AISC 360 Direct Analysis (on page 2352)
D1.A.1 General Comments on Design as per AISC Unified Code

Both Allowable Stress Design and Load Resistance Factor Design methods are implemented in STAAD. The selection of the method can be done through the METHOD parameter explained in the parameter list. This Unified Code allows the designer to design the member as per LRFD as well as ASD method.

**Design for Strength Using Load and Resistance Factor Design (LRFD)**

Design according to the provisions for Load and Resistance Factor Design (LRFD) satisfies the requirements of the AISC 360 Unified Code Specification, when the design strength of each structural component equals or exceeds the required strength determined on the basis of the LRFD load combinations.

Design shall be performed in accordance with Equation B3-1 of the Code:

\[ R_u \leq \varphi R_n \]

where

- \( R_u \) = required strength (LRFD)
- \( R_n \) = nominal strength
- \( \varphi \) = resistance factor
- \( \varphi R_n \) = design strength

**Design for Strength Using Allowable Strength Design (ASD)**

Design according to the provisions for Allowable Strength Design (ASD) satisfies the requirements of the AISC 360 Unified Code Specification when the allowable strength of each structural component equals or exceeds the required strength determined on the basis of the ASD load combinations.

Design shall be performed in accordance with Equation B3-2 of the Code:

\[ R_a \leq R_n / \Omega \]

where

- \( R_a \) = required strength (ASD)
- \( R_n \) = nominal strength
- \( \Omega \) = safety factor
- \( R_n / \Omega \) = allowable strength

D1.A.2 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, non-compact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro is capable of determining the section classification for the standard shapes and design accordingly.

The Section Classification is done as per section B4 and Table B4.1, for Stiffened and Un-Stiffened Elements of a section.

D1.A.3 Member Properties

For specification of member properties of standard American steel sections, the steel section library available in STAAD.Pro may be used. The syntax for specifying the names of built-in steel shapes is described in the next section.
D1.A.4 Built-in Steel Section Library

The following sections describe specification of steel sections from the AISC Steel Tables.

Related Links

• G.6.2 Built-In Steel Section Libraries (on page 2325)

D1.A.4.1 AISC Steel Table

Almost all AISC steel shapes are available for input. Following are the descriptions of all the types of sections available:

Wide Flanges (W shapes)

All wide flange sections as listed in AISC are available the way they are written, e.g., W10X49, W21X50, etc.

20 TO 30 TA ST W10X49
33 36 TA ST W18X86

C, MC, S, M, HP Shapes

The above shapes are available as listed in AISC (9th Edition) without decimal points. For example, C8X11.5 will be input as C8X11 and S15X42.9 will be input as S15X42, omitting the fractional portion of the weight past the decimal.

Note: Exception: MC6X151 for MC6X15.1 and MC6X153 for MC6X15.3.

10 TO 20 BY 2 TA ST C15X40
1 2 TA ST MC8X20

Double Channels

Back to back double channels, with or without spacing between them, are available. The letter D in front of the section name will specify a double channel.

21 22 24 TA D MC9X25
55 TO 60 TA D C8X18

Front to front channel steel sections can be defined by using the FR (Double Channel – Front to Front) option and any spacing required between the channels are specified with the SP designation.

61 62 TABLE FR C4X5 SP 0.5

Note: The SP parameter is optional, but if it is not set, the section will not be assumed to be a closed box for torsional calculations.

Angles

Angle specifications in STAAD.Pro are different from those in the AISC manual. The following example illustrates angle specifications.

\[ L_{40356} = L \times 3-1/2 \times 3/8 \]
Similarly, $L_{505010} = L 5 \times 5 \times 5/8$ and $L_{904016} = L 9 \times 4 \times 1$

At present, there are two ways to define the local y and z-axes for an angle section. To make the transition from the AISC Manual to the program data easy, the standard section for an angle is specified:

$51 \ 52 \ 53 \ TA \ ST \ L40356$

This specification has the local z-axis (i.e., the minor axis) corresponding to the Z-Z axis specified in the steel tables. Many engineers are familiar with a convention used by some other programs in which the local y-axis is the minor axis. STAAD provides for this convention by accepting the command:

$54 \ 55 \ 56 \ TA \ RA \ L40356$

**Note:** RA denotes reverse angle

**Double Angles**

Short leg back to back or long leg back to back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle either LD or SD will serve the purpose.

$14 \ TO \ 20 \ TA \ LD \ L35304 \ SP \ 0.5$

Long leg back to back L3-1/2x3x1/4 with 0.5 space.

$23 \ 27 \ TA \ SD \ L904012$

Short leg back to back L 9x4x3/4

**Tees**

Tees are not input by their actual names, as they are listed in the AISC manual, but instead by designating the beam shapes (W and S) from which they are cut. For example

$1 \ 2 \ 5 \ 8 \ TA \ T \ W8X24$

Tee cut from W8x24, or a TW4x12.

**Pipes**

Two types of specifications can be used for pipe sections. Pipe sections listed in the AISC manual can be specified as follows.

$5 \ TO \ 10 \ TA \ ST \ PIPX20$

**PIPX20** = extra-strong pipe 2" in dia.

Pipe symbol - 10 times diameter in inches only portion before decimal

Strength spec -

Where the strength spec is one of the following:

- S = standard
- X = extra-strong
- D = double extra-strong

General pipe sections may be input by their outer and inner diameters. For example,

$1 \ TO \ 9 \ TA \ ST \ PIPE \ ØD \ 2.0 \ ID \ 1.875$

Indicates a pipe with an outer diameter of 2.0 and inner diameter of 1.875 in current input units.
Round HSS

Round hollow structural sections listed in the AISC manual can be specified as follows.

| 5 TO 10 | HSS16X0.438 |
| 11 TO 12 | HSS14X0.5_A1085 |

Indicates members 5 through 10 as 16” outer diameter and a nominal wall thickness of 7/16” (0.4375”). Members 11 and 12 have a 14” outer diameter and a 1/2” nominal wall thickness and use A1085 grade material.

**Notes:**

- AISC codes require that the design of HSS sections use section properties calculated based on a reduced wall thickness. The STAAD.Pro database values incorporated a design thickness of 0.93× the nominal thickness.
- HSS sections using A1085 grade material per the AISC 360-16 and 360-10 codes are designed using section properties with the full nominal wall thickness. Thus, a separate set of tables is available to specify this material grade as these use different section properties.

Tubes

Two types of specifications can be used for tube sections. Tube sections from the AISC tables can be specified as follows.

| 5 TO 10 | TA ST TUB120808 |

Indicates a tube that has a height of 8, a width of 6, and a wall thickness of 0.5 in the current input units.

**Note:** Member Selection cannot be performed on tubes specified by their dimensions. Only code checking can be performed on these sections.

Rectangular HSS

Rectangular hollow structural sections listed in the AISC manual can be specified as follows.

| 5 TO 10 | HSST14X10X0.313 |
| 11 TO 12 | HSST6X3X0.375_A1085 |

Indicates members 5 through 10 as a 14” x 10” tube with a nominal wall thickness of 5/16” (0.3125”). Members 11 and 12 are a 6” x 3” tube with a 3/8” nominal wall thickness and use A1085 grade material.
**D1.A.4.2 Welded Plate Girders**

The AISC welded plate girder shapes (pages 2-230 and 2-231 – AISC 9th edition) are available in the Steel Section library of the program.

<table>
<thead>
<tr>
<th>Number</th>
<th>Type</th>
<th>Material</th>
<th>Section Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 10</td>
<td>TA</td>
<td>ST</td>
<td>B612010</td>
</tr>
<tr>
<td>15 16</td>
<td>TA</td>
<td>ST</td>
<td>B682210</td>
</tr>
</tbody>
</table>

**Nomenclature**

The nomenclature used for welded plate girders includes:
- **Built-up section symbol**: B
- **Nominal Depth in inches**: 61 20 10
- **Thickness of flange in inches X 10 (Only use portion before decimal point)**
- **Nominal flange Width (inches)**

**D1.A.4.3 Castellated Beams Section Sizes**

STAAD.Pro incorporates the non-composite castellated beam tables.

All castellated beams are listed with their nominal depth and their nominal weight per foot (i.e., similar to the nomenclature used for American wide flange shapes).

<table>
<thead>
<tr>
<th>Castellated beam symbol</th>
<th>Nominal weight per foot in lbs</th>
<th>Nominal depth in inches</th>
</tr>
</thead>
<tbody>
<tr>
<td>CB15X33</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Tip**: You can click View Table in the Section Profile Tables dialog to view the properties table, which includes the root beam profile size used for each castellated beam.

<table>
<thead>
<tr>
<th>Number</th>
<th>Type</th>
<th>Material</th>
<th>Section Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 TO 20</td>
<td>TA</td>
<td>ST</td>
<td>CB15X33</td>
</tr>
<tr>
<td>2</td>
<td>TA</td>
<td>ST</td>
<td>CB27X40</td>
</tr>
</tbody>
</table>

**Related Links**
- [D1.B.2 Castellated Beams](on page 1422)

**D1.A.5 Limit States**
D1.A.5.1 Axial Tension

The criteria governing the capacity of tension members are based on:

- Tensile Yielding in Gross Section and
- Tensile Rupture of Net Section.

The limit state of yielding in the gross section is intended to prevent excessive elongation of the member, and the corresponding check is done as per section D2-(a) of the code.

The second limit state involves fracture at the section with the minimum effective net area, and the corresponding check is done as per section D2-(b) of the code.

STAAD.Pro calculates the tension capacity of a given member based on these two limit states.

The Net Section Area may be specified by the user through the use of the parameter NSF (see D1.B.1.2 Design Parameters (on page 1409)). The Effective Net Area of tension members can be determined by using the Shear Lag Factor. You can also input the shear lag factor through the use of the parameter SLF.

D1.A.5.2 Axial Compression

The Design Compressive Strength (LRFD), $\phi_c \times P_n$, and the Allowable Compressive Strength (ASD), $P_n / \Omega_c$, are calculated by the program.

The Nominal Compressive Strength, $P_n$, shall be the minimum value obtained according to the Limit States of:

- Flexural Buckling,
- Torsional Buckling, and
- Flexural-Torsional Buckling.

The Nominal Compressive Strength, $P_n$, for a particular member is calculated by STAAD.Pro according to the procedure outlined in Chapter E, section E3 to E5, of the unified code specifications. For slender elements, the procedure described in section E7 is used.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY and KZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD.Pro immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

D1.A.5.3 Flexural Design Strength

The Design Flexural Strength (LRFD), $\phi_b \cdot M_n$, and the Allowable Flexural Strength (ASD), $M_n / \Omega_b$, are being calculated by the program.

The Nominal Flexural Strength, $M_n$, is determined according to Sections F2 through F12 of unified code specifications, for different types of rolled sections.

Doubly symmetric, I-shaped sections with slender flanges are also checked, as per Section F3 of AISC 360-10.

The Nominal Flexural Strength of a member is determined by the limit states of Yielding (Y), Lateral-Torsional Buckling (LTB), Flange Local Buckling (FLB), Web Local Buckling (WLB), Tension Flange Yielding (TFY), Leg Local Buckling (LLB), and Local Buckling (LB).
The program internally calculates the Lateral-Torsional Buckling Modification Factor \( C_b \) for non-uniform moment diagrams when both ends of the unsupported segment are braced. The purpose of this factor is to account for the influence of the moment gradient on lateral-torsional buckling.

To specify laterally unsupported length, the parameter UNF can be used, by default which takes the value of the member length.

**D1.A.5.4 Design for Shear**

The Design Shear Strength (LRFD), \( Q_v \times V_n \), and the Allowable Shear Strength (ASD), \( V_n / \Omega_v \), are calculated by the program, as per section G2 of the unified code specifications.

The Nominal Shear Strength, \( V_n \), of un-stiffened or stiffened webs, is calculated taking care of limit states of shear yielding and shear buckling. The sections G4 to G7 of the code specifications are used to evaluate Nominal Shear Strength, \( V_n \) for different types of rolled sections.

Shear Capacity Along the Major Axis

For determining shear capacity, \( V_c \) in the major axis Clause G2, G3, G4 are followed based on the different shapes.

**Note:** For user-defined sections the value of shear area will be used instead of the term ‘Aw’ in the equation in the above-mentioned clauses.

Shear Capacity Along the Weak Axis

The nominal shear strength, \( V_n \) for each shear resisting element in doubly symmetric and singly symmetric shapes loaded in the weak axis (minor axis, or along the flanges) without torsion in is determined per AISC 360-16 G6 as follows:

\[
V_n = 0.6F_y b_f t_f C_{v2} \tag{G6-1}
\]

The use of the section dimensions\[1\] \( b_f \& t_f \) in the above equation will be based on the section profile type. The following table lists the value of the terms \( b_f \), \( t_f \), \( k_v \), \( C_{v2} \) that are used in the calculation of shear capacities for various section profile shapes, when subject to shear along the z-axis of the section.

<table>
<thead>
<tr>
<th>Shape</th>
<th>Use of terms ( b_f t_f ) in eqn G6-1</th>
<th>Dims used to calculate shear buckling coeff. ( C_{v2} ) (Ref Cl G.2)</th>
<th>Ceoff ( k_v ) used to calculate ( C_{v2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Built Up Box</td>
<td>( b_f = 2.0 \times (B - 2 \times \text{wall thickness}) &amp; t_f = \text{flange thickness} )</td>
<td>( h = (B - 2 \times \text{wall thickness}) ) &amp; ( t = \text{flange thickness} )</td>
<td>5.0</td>
</tr>
<tr>
<td>HSS Box</td>
<td>( b_f = 2.0 \times \text{Width between fillets} &amp; t_f = \text{wall thickness} )</td>
<td>( h = \text{Width between fillets} ) &amp; ( t = \text{wall thickness} )</td>
<td>5.0</td>
</tr>
<tr>
<td>Rolled/Channel</td>
<td>( b_f = 2.0 \times \text{Flange width} &amp; t_f = \text{flange thickness} )</td>
<td>( h = \text{Flange Width} &amp; t = \text{Flange thickness} )</td>
<td>1.2</td>
</tr>
<tr>
<td>Rolled/Built Up I</td>
<td>( b_f = 2.0 \times \text{Flange width} &amp; t_f = \text{flange thickness} )</td>
<td>( h = \text{Flange Width} &amp; t = \text{Flange thickness} )</td>
<td>1.2</td>
</tr>
</tbody>
</table>
### Use of terms \( b_f t_f \) in eqn G6-1

<table>
<thead>
<tr>
<th>Shape</th>
<th>Use of terms ( b_f t_f ) in eqn G6-1</th>
<th>Dims used to calculate shear buckling coeff. ( C_{v2} ) (Ref Cl G.2)</th>
<th>Coeff ( k_v ) used to calculate ( C_{v2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>I With Cover Plate(s)[2]:</td>
<td>( B_f \times t_f = 2.0 \times \text{Flange width} \times \text{flange thickness} + \text{width of top cover plate} \times \text{thickness of top plate} + \text{width of bottom plate} \times \text{thickness of bottom plate} )</td>
<td>( h = \text{Flange Width} \times t = \text{Flange thickness for the base I shape} ) &amp; ( h = \text{width of plate and } t = \text{thickness of plate} )</td>
<td>1.2</td>
</tr>
<tr>
<td>L- Section</td>
<td>( b_f = \text{Length of leg along the X-axis} &amp; t_f = \text{thickness} )</td>
<td>( h = \text{Length of leg along the X-axis} ) &amp; ( t = \text{thickness} )</td>
<td>1.2</td>
</tr>
<tr>
<td>T - Section</td>
<td>( b_f = \text{Flange width} ) &amp; ( t_f = \text{flange thickness} )</td>
<td>( h = \text{Flange width} ) ( (b_f) ) &amp; ( t = \text{flange thickness} )</td>
<td>1.2</td>
</tr>
<tr>
<td>Solid bar / rod[3]</td>
<td>( B_f \times t_f = 0.5 \times \text{section area} )</td>
<td>1.0</td>
<td>-</td>
</tr>
</tbody>
</table>

### Notes:

1. For the shear in the weak axis, the value is determined using \( B_f \) and \( T_f \) alone. \( AZ \) cannot be used.
2. The shear capacities of the base I section and the top and/or bottom cover plates are evaluated separately using the dims shown and a \( k_v = 1.2 \). These values are then added to get the final section shear strength.
3. For solid rectangular bars and rods, half the total section area is assumed to resist the shear along the vertical axis and the other half to resist the shear along the horizontal axis. This could lead to conservative results in some cases.

### Related Links

- [V. AISC 360-16 Shear Strong Axis](on page 3866)
- [V. AISC 360-16 Shear Weak Axis](on page 3881)

#### D1.A.5.5 Design for Combined Forces

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by sections H1 and H3. These interaction formulas cover the general case of biaxial bending combined with axial force and torsion. They are also valid for uniaxial bending and axial force.

#### D1.A.5.6 Design for Torsion

Stresses due to torsion in non-HSS sections are considered per AISC 360-10 Section H3.3. This section states that the available torsional strength for non-HSS members shall be the least value obtained according to the limit states of yielding under normal stress, shear yielding under shear stress, or buckling:

\[
\phi_t = 0.9 \quad (LRFD), \quad \Omega_t = 1.67 \quad (ASD)
\]

For the limit state of yielding under normal stress (H3-7):

\[
F_n = F_y
\]

For the limit state of shear yielding under shear stress (H3-8):
The calculation of $F_v$ and $F_n$ is based on AISC Design Guide 9 *Torsional Analysis of Structural Steel Members* (DG-9). In general terms, in case of shear stress, $F_v$ will comprise of components of shear stress due to shear about both axes, warping shear stress and shear stress due to pure torsion. In case of normal stress $F_n$, stress due to axial force and stress due to flexure about both axes is considered. For some sections, like Single Angles and Tees, the component due to warping is negligible with respect to stress for pure torsion (Ref. Section 4.2 and 4.3 of Design Guide 9).

**Notes**

- STAAD.Pro will perform these torsion design checks when the **TORSION** parameter has been set to 1 (these are *not* checked by default).
- When torsion checks are performed, TRACK 3 output may be used to provide detailed torsion design output for Design Guide 9 checks.
- The torsion checks per Design Guide 9 require additional analysis to calculate $\Theta$ (rotation of the element due to applied torsion; refer to the following sections) at the 13 design segments along the member. Thus there is a performance cost for each torsion check. Therefore, it is recommended that torsion checks only be performed on the necessary members (rather than all members).

**Pure Torsional Shear Stress**

These shear stresses are always present in the cross-section of a member subjected to torsional moment. They are in plane shear stresses which vary linearly along the thickness of an element.

\[ \tau_t = G t \Theta' \]

where

- $\tau_t$ = the pure torsional shear stress at the element edge
- $G$ = the shear modulus of elasticity of steel
- $t$ = thickness of the element
- $\Theta'$ = first derivative of rotation expressed as a function of local $x$ (distance from start end to point where rotation is calculated)

In the case of an element with a rectangular cross section:

\[ \tau_t = T_u t/J \]

In the case of a hollow circular element or a pipe section with an inner radius of $R$:

\[ \tau_t = T_u R/J \]

In the case of a tube section:

\[ \tau_t = T_u/(2bht) \]

where

- $T_u$ = the total torsional moment action at any location along the beam

**Shear Stress Due to Warping**

When warping in a member is restrained, in plane shear stresses are developed which are constant along thickness of the element but vary along the length of the element.

\[ \tau_{ws} = -E S_{ws} \Theta''/t \]

where

\[ F_v = 0.6 F_y \]
where
\[ \tau_{ws} = \text{shear stress at a point, } s, \text{ due to warping} \]
\[ E = \text{the modulus of elasticity of steel} \]
\[ t = \text{thickness of the element} \]
\[ S_{ws} = \text{warping statical moment at a point, } s \]
\[ \Theta''' = \text{third derivative of rotation expressed as a function of local } x \text{ (distance from start end to point where rotation is calculated)} \]

**Note:** The shear stress due to warping is neglected for angle, tee, tube, or pipe sections.

Normal Stress Due to Warping

When warping in a member is restrained, direct stress acting perpendicular to the cross-section of the element is generated. These stresses are constant along the cross-section but vary along the length of the member.

\[ \sigma_{ns} = EW_{ns}\Theta'' \]

where
\[ \sigma_{ns} = \text{normal stress at a point, } s, \text{ due to warping} \]
\[ E = \text{the modulus of elasticity of steel} \]
\[ W_{ns} = \text{normalized warping function at a point, } s \]
\[ \Theta'' = \text{second derivative of rotation expressed as a function of local } x \text{ (distance from start end to point where rotation is calculated)} \]

**Note:** Point s refers to a point on the cross section area of a particular section as explained in Section 3.2.2 of Design Guide 9.

Combined Stresses due to Axial, Bending, and Torsional Stresses

Under section 4.6 of Design Guide 9, the combined stress in a section due all the stresses as explained in sections 4.1, 4.2, 4.3, and 4.4 is

\[ f_n = \sigma_a + \sigma_{bz} + \sigma_{by} + \sigma_s \]
\[ f_t = \tau_{sz} + \tau_{sy} + \tau_t + \tau_{ws} \]

These stresses are calculated at 13 sections along the beam length.

Considered Loads

Only member loads of the following types are considered in these checks:

- concentrated torque (moment about local x axis)
- concentrated force eccentric from the member shear center
- uniformly distributed torque (full or partial)
- uniformly distributed force eccentric to the member shear center
- end torques (only considered when end supports are fixed)

Linearly varying torque is *not* considered in the torsion checks. Joint loads are also *not* considered in the torsion checks.

The boundary conditions for torsional analysis and the method to calculate rotation, \( \Theta \), and its derivatives are used as described in DG-9.

STAAD.Pro calculates the stresses due to flexure, pure torsion, and warping torsion at 13 different sections along the member length. The total stress is the vector summation at each location.
D1.A.5.7 Design of Web-Tapered Members

AISC 360 05/10 specifications have been incorporated into STAAD.Pro to perform code checking on web tapered wide flange, square, and round shapes.

**Note:** Member selection cannot be performed on web-tapered members. That is, SELECT ALL, SELECT OPTIMIZE, and PROFILE are not applicable to web-tapered members.

The section properties used for web-tapered members are based on interpolated values between the start and depths of the member. Similar interpolation of values from start and end values is done for square and round tapered members.

**Related Links**

- **TR.20.3 Tapered Member Specification** (on page 2468)

D1.A.5.8 Design of I-Section with Cover Plates

STAAD.Pro can design I-sections with cover plates per AISC 360-05 and AISC 360-10.

According to B4.2(c) (p.16.1-15) of the AISC 360-05 specification, the cover plate is taken as a stiffened element and the width of a flange plate in a built-up section is taken between as the distance between lines of fasteners or welds. For the compression flange, the appropriate classification in Table B4.1 is that of case 12, which explicitly includes “flange cover plates and diaphragm plates between lines of fasteners or welds.” The section classification limits for this case are:

\[
\lambda = \frac{b}{t}
\]

\[
\lambda_p = 1.12 \sqrt{\frac{E}{F_y}}
\]

\[
\lambda_r = 1.40 \sqrt{\frac{E}{F_y}}
\]

where

- \( b \) = width of the cover plate
- \( t \) = thickness of the cover plate

Similarly, according to B4.2(c) of the AISC 360-10 specification, the cover plate is taken as a stiffened element and the width of the flange plate in a built-up section is taken as the distance between lines of fasteners or welds. For the compression flange, case 7 is used.

\[
\lambda = \frac{b}{t}
\]

\[
\lambda_r = 1.40 \sqrt{\frac{E}{F_y}}
\]

For flexure, case 18 is used.

\[
\lambda = \frac{b}{t}
\]
\[ \lambda_p = 1.12 \sqrt{\frac{E}{F_y}} \]

\[ \lambda_r = 1.40 \sqrt{\frac{E}{F_y}} \]

The section classification is included in the STAAD.Pro output listed as UNSTIFFENED / STIFFENED elements of the shape, along with the classification limits calculated as above and the element slenderness ratio, \( \lambda \).

In addition to the limit states described elsewhere in this section, I-sections with cover plates have the following considerations for code checking per AISC 360-05/10:

- For Section E4, the program determines if the section is doubly or singly symmetric. Based on this, E.4-4 or E.4-5 is used for the compression code check, respectively. An I-section with the same cover plate top and bottom remains doubly symmetric. An I-section with differing top and bottom plates or only a top or a bottom cover plate is singly symmetric.

- For lateral-torsional buckling calculations, the values of \( r_t \) and \( a w \) are calculated as described in F4.2 for the effective radius of gyration in section ii.

- For shear, the minimum of all b/t ratios - including the flanges and top and bottom cover plates - is used for the web slenderness in the horizontal direction.

- For shear, the cover plates are checked for tension field action.

- The cover plates are considered in the seismic classification of the section.

It is assumed that the section classifications are for I-section flanges, web, and cover plates but not for parts of elements (i.e., outstanding flange).

**Note:** The following two items are not checked for I-sections with cover plates:

- The effects of the cover plates for flange local buckling of the section. Only the I-section flanges are considered in the FLB calculation.
- AISC Design Guide 9 is not incorporated.

**D1.A.6 Design Parameters**

Design per AISC 360-05, 360-10, and 360-016 (Unified) specifications is requested by using the CODE parameter. Other applicable parameters are summarized in the following Table. These parameters communicate design decisions from the engineer to the program and thus allow you to control the design process.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.
Table 90: AISC 360-05, 360-10, and 360-16 Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>AISC UNIFIED</td>
<td>Used to designate this code (default is the 2016 edition).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CODE AISC UNIFIED (2016)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CODE AISC UNIFIED 2010</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CODE AISC UNIFIED 2005</td>
</tr>
<tr>
<td>ALH</td>
<td>0.5</td>
<td>Distance of applied point torsion from start of member as a fraction of member length. Represented by “α” in the torsional case options in Appendix B of AISC Design Guide 9. To be used with TND values of 3, 6, or 9. (0&lt;ALH&lt;1). Set TORSION 1 to enable Torsion check as per DG9.</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>See Note 9 below.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0 = design at start and end nodes and those locations specified by the SECTION command.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes.</td>
</tr>
<tr>
<td>BRC</td>
<td>1</td>
<td>Specifies the bracing type for the member used for seismic provision checks:</td>
</tr>
<tr>
<td>(AISC 360-05 and 360-10 only)</td>
<td></td>
<td>1 = Relative bracing</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = Nodal bracing</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = deflection check based on the principle that maximum deflection is of the cantilever type (see D1.B.1.2 Design Parameters (on page 1409))</td>
</tr>
<tr>
<td>CB</td>
<td>2 1.0</td>
<td>Coefficient $C_b$ per Chapter F. If $C_b$ is set to 0.0, it will be calculated by the program. Any other value will be directly used in the design. See Note 2 below.</td>
</tr>
<tr>
<td>CSPACING</td>
<td>12 in</td>
<td>Spacing between connectors in current length units. Refer to Section E6.1 and E6.2 of AISC 360.</td>
</tr>
<tr>
<td>DFF</td>
<td>none (mandatory for deflection check)</td>
<td>“Deflection Length” / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of “Deflection Length” (see D1.B.1.2 Design Parameters (on page 1409))</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot; (see D1.B.1.2 Design Parameters on page 1409))</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000.0 mm</td>
<td>Maximum allowable depth for member selection.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 mm</td>
<td>Minimum allowable depth for member selection.</td>
</tr>
<tr>
<td>DUCT (AISC 360-16 only)</td>
<td>0</td>
<td>The ductile category of the member as per AISC 341-16: 0 = non-ductile, 1 = moderately ductile, 2 = highly ductile</td>
</tr>
<tr>
<td>FLX</td>
<td>1</td>
<td>Parameter for specifying the lateral-torsional restraint condition for a single angle. Refer to Section F10 of AISC 360-05, 360-10, and 3601-16. 1 = Member does not have continuous lateral-torsional restraint along the length. 2 = Member has continuous lateral-torsional restraint along the length. 3 = Lateral-torsional restraint is provided at the point of maximum moment only.</td>
</tr>
<tr>
<td>FRM (AISC 360-05 and 360-10 only)</td>
<td>0</td>
<td>Specifies the seismic force-resisting system used in seismic provision checks: 0 = Ordinary Moment Frame (OMF) 1 = Intermediate Moment Frame (IMF) 2 = Special Moment Frame (SMF)</td>
</tr>
<tr>
<td>FU</td>
<td>400 MPa</td>
<td>Ultimate strength of steel.</td>
</tr>
<tr>
<td>FYLD</td>
<td>250 MPa</td>
<td>Yield strength of steel. The program considers a valid range of input values between 10 ksi - 100 ksi (69 MPa - 689 MPa).</td>
</tr>
<tr>
<td>KX</td>
<td>1.0</td>
<td>K value for flexural-torsional buckling.</td>
</tr>
</tbody>
</table>

AISC 360-16: This value is ignored if SGR specified other than 0.

AISC 360-16: This value is ignored if SGR specified other than 0.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| INTERACTION   | 0            | Directs the program which interaction equations to check per section H1:  
                |              | 0 = Checks both H1.1 and H1.3 and reports the lower ratio as critical  
                |              | 1 = Check both H1.1 and H1.3 and reports the higher ratio as critical  
                |              | 2 = Always checks per H1.1 even when H1.3 may be applicable  
                |              | 3 = Checks H1.3 in lieu of H1.1 when applicable |
| KY            | 1.0          | Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis. |
| KZ            | 1.0          | Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis. |
| LBRC (AISC 360-16 only) | 1 | Type of flange lateral bracing:  
                |              | 0 = none  
                |              | 1 = panel bracing  
                |              | 2 = point bracing  
                |              | 3 = special bracing  
                |              | Used to calculate bracing requirements as per seismic provisions in AISC 341-16. |
| LEG           | 0            | This parameter is meant for plain angles (Section E5).  
                |              | 0 = The angle is connected by the longer leg.  
<pre><code>            |              | 1 = The angle is connected by the shorter leg. |
</code></pre>
<p>| LX            | Member Length | Length for flexural-torsional buckling. See Note 8 below. |
| LY            | Member Length | Length to calculate slenderness ratio for buckling about local y-axis. |
| LZ            | Member Length | Length to calculate slenderness ratio for buckling about local z-axis. |
| MAIN          | 200          | Allowable slenderness limit for compression members. |
| METHOD        | LRFD         | Used to specify LRFD or ASD design methods. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MTYP</strong></td>
<td>1</td>
<td>Specifies whether the member is a beam or column. Used for seismic provisions checks.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Beam</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = Column</td>
</tr>
<tr>
<td></td>
<td></td>
<td>For AISC 360-16 only:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 = Beam-column</td>
</tr>
<tr>
<td><strong>NBRC</strong></td>
<td>1</td>
<td>Number of braced points within the span. Represented by “n” in Appendix 6.3.2(a) of AISC 360-16. Required for Seismic Provisions.</td>
</tr>
<tr>
<td>(AISC 360-16 only)</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>NSF</strong></td>
<td>1.0</td>
<td>Net Section Factor for tension members, equal to $A_n/A_g$, used to account for reduction in section used for tension checks (clause B 4.3b.) combined with the SLF parameter to determine the rupture strength. (see also SLF parameter)</td>
</tr>
<tr>
<td><strong>PROFILE</strong></td>
<td></td>
<td>Used in member selection. Refer to [TR.48.1 Parameter Specifications](on page 2851) for details.</td>
</tr>
<tr>
<td><strong>RATIO</strong></td>
<td>1.0</td>
<td>Permissible ratio of actual load to allowable strength.</td>
</tr>
<tr>
<td><strong>SEISMIC</strong></td>
<td>0</td>
<td>Specifies whether the seismic provisions according to 341-05/10/16 should be checked:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 = Do not check seismic provisions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Check seismic provisions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>See [D.1.A.9 Seismic Provision Checking per AISC 341](on page 1388) for details.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SGR (AISC 360-16 only)</td>
<td>Select ASTM steel grades:</td>
<td></td>
</tr>
</tbody>
</table>
| | Custom  
1 = A36  
2 = A53 Gr.B  
3 = A500 Gr.B (HSSRect)  
4 = A500 Gr.B (HSSRound)  
5 = A500 Gr.C (HSSRect)  
6 = A500 Gr.C (HSSRound)  
7 = A501 Gr.A  
8 = A501 Gr.B  
9 = A529 Gr.50  
10 = A529 Gr.55  
11 = A709 Gr.36  
12 = A1043 Gr.36  
13 = A1043 Gr.50  
14 = A572 Gr.42  
15 = A572 Gr.50  
16 = A572 Gr.55  
17 = A572 Gr.60  
18 = A572 Gr.65  
19 = A618 Gr.I(a)/Gr.I(b)/Gr.II  
20 = A618 Gr.III  
21 = A709 Gr.50  
22 = A709 Gr.50S  
23 = A709 Gr.50W  
24 = A913 Gr.50  
25 = A913 Gr.60  
26 = A913 Gr.65  
27 = A913 Gr.70  
28 = A992  
29 = A588  
30 = A847  
31 = A1085 | The yield stress and ultimate stress will be auto-calculated based on the grade selected. Note that any SGR value greater than 0 will take priority when calculating the yield stress and ultimate stress over any supplied FYLD and FU value. |

**Note:** “HSS Rectangle A1085” and “HSS Round A1085” profiles steel grade will always be considered as A1085 irrespective of any value assigned to SGR.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SLF</td>
<td>1.0</td>
<td>Shear Lag Factor, value “U” normally taken from table D3.1, combined with the NSF parameter to determine the net effective area used to calculate the section rupture strength. (see also NSF parameter)</td>
</tr>
</tbody>
</table>
| SNUG           | 1             | Type of connection for the built-up members:  
|                |               | 0 = Welded or pretensioned bolts  
|                |               | 1 = Bolted snug-tight |
| SOE            | 0             | Second Order Effects have been considered in analysis forces or not:  
| (AISC 360-16 only) |               | 0 = Have not been considered  
|                |               | 1 = Have been considered  
|                |               | By default, Second Order Effects are not considered in the analysis forces. This is related to Torsion checks as per DG9. Set TORSION 1 to enable Torsion check as per DG9. |
| STFB           | 0.0           | Stiffener width for one-sided web stiffeners, twice the individual stiffener width for pairs of stiffeners. Represented by “bs” in Appendix 6.3.2(a) of AISC 360-16. Required for Seismic Provisions. |
| STFT           | 0.0           | Thickness of web stiffeners. Represented by “tst” in Appendix 6.3.2(a) of AISC 360-16. Required for Seismic Provisions. |
| STIFF          | Member Length or depth of beam, whichever is greater | Spacing of stiffeners for plate girder design. |
| STP            | 1.0           | Section Type used for design  
|                |               | 1 = Rolled section  
|                |               | 2 = Welded section  
|                | Note: If a UPT Wide flange section with different top & bottom flange dimensions have been specified for a member, the AISC360-16 module will ignore the value of STP and consider the section as a Welded/Built-Up section. |
### Parameter Name | Default Value | Description
--- | --- | ---
TBRC | (AISC 360-16 only) | Type of torsional bracing:
- 0 = None
- 1 = continuous bracing
- 2 = point bracing
- 3 = special bracing

Used to calculate bracing requirements as per seismic provisions in AISC 341-16.

TFA | (AISC 360-16 only) | 0
Tension field action to be considered in shear design:
- 0 = do not consider
- 1 = consider

TMAIN | 300 | Allowable slenderness limit for tension members.

TND | (AISC 360-16 only) | 1
Torsion loading and end condition as in Table in Appendix C.4 of AISC Design Guide 9:
- 1 = Equal concentrated end torques. Both ends free.
- 2 = Equal concentrated end torques. Both ends fixed.
- 3 = Concentrated Torque. Both ends pinned.
- 4 = Uniformly distributed torque. Both ends pinned.
- 6 = Concentrated Torque. Both ends fixed.
- 7 = Uniformly distributed torque. Both ends fixed.
- 9 = Concentrated Torque. One end fixed, another end free.
- 10 = Partial uniformly distributed torque. One end fixed, another end free.
- 12 = Uniformly distributed torque. One end fixed, another end pinned.

The number corresponds to the case number of the case chart in Appendix B of DG9. Set TORSION 1 to enable Torsion check as per DG9.

TORSION | (AISC 360-10 and 360-16 only) | 0
Specifies design for torsion per AISC Design Guide 9. See D1A.5.6 Design for Torsion (on page 1374)
- 0 = Do not perform torsion checks
- 1 = Perform torsion checks

**Note:** When torsion checks are performed, TRACK 3 output may be used to provide detailed torsion design output for Design Guide 9 checks.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| TRACK          | 0             | Specifies the amount of detail included in design output  
|                |               | 0 = Suppress all member capacities  
|                |               | 1 = Print all member capacities  
|                |               | 2 = Print full member design details |
| UNB            | Member Length | Unsupported length of the bottom flange for calculating flexural strength. Will be used only if compression is in the bottom flange. See Note 3 below. |
| UNL            | Member Length | Unsupported length of left extreme flange for LTB that will be used as lateral-torsional buckling length for the section with the vertical axis as major principal axis and where the left extreme fiber is in compression. If member is assigned with any value of UNL and FLX=2 concurrently, LTB length of the member will be treated as zero. |
| UNR            | Member Length | Unsupported length of right extreme flange for LTB that will be used as lateral-torsional buckling length for the section with the vertical axis as major principal axis and where the right extreme fiber is in compression. If member is assigned with any value of UNR and FLX=2 concurrently, LTB length of the member will be treated as zero. |
| UNT            | Member Length | Unsupported length of the top flange for calculating flexural strength. Will be used only if compression is in the top flange. See Note 3 below. |
| WTYP (AISC 360-05 and 360-10 only) | 0 | Weld type for HSS per Sect. B3.12 (AISC 360-05) or Sect. B4.12 (AISC 360-10):  
|                |               | 0 = Electric resistance welding  
|                |               | 1 = Submerged arc welding  
|                |               | For HSS Rectangle and Round profiles from AISC databases, the weld type will always be determined based on profile table (grade A1085 or not). WTYP will have no effect on these. For any other hollow profiles (pipe, tube, box, etc.) from AISC databases, all hollow profiles from any other country databases, and User Provided tables, the weld type must be specified using the WTYP parameter. |

**Notes**

1. For the AISC 360 unified code, an angle is automatically checked for geometric axis bending (in addition to principal axis bending) provided one of the following conditions is met:
   
   i. The FLX parameter is set to 2 for that member. The section could be an equal or an unequal legged angle.
ii. The angle is equal-legged, has bending moment only about one of its geometric axes, and is not subjected to axial compression.

The AXIS parameter is only used by the deprecated AISC 360-05 code checking method (CODE AISC UNIFIED OLD). If this code is used, then AXIS 1 specifies design based on principle axes, where AXIS 2 specifies design based on geometric axes.

2. Non-default values of CB must be re-entered before every subsequent CHECK CODE or SELECT command.

3. Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used).

4. For a description of the deflection check parameters DFF, DJ1, DJ2 see the Notes section of D1.B.1.2 Design Parameters (on page 1409) of this manual.

5. NSF is the Net Section Factor as used in most of the steel design codes in STAAD.Pro. It is defined as the Ratio of “Net cross section area” / “Gross section area” for tension member design. The default value is 1.0. For the AISC 360 code, it is described in section D3.2.

6. SLF is the Shear Lag Factor, as used in Section D.3.3 of the AISC 360-05 code. This factor is used to determine the effective net area by multiplying this factor with net area of the cross section. Please refer to Table D3.1 of the 360 code for a list of acceptable SLF values. In STAAD.Pro, the default value for SLF is 1.0. The effective net area is used to determine the tensile strength for tensile rupture in the net section, as per equation D.2.2.

7. To summarize, the “Gross Area” (Ag) is multiplied by NSF to get the “Net Area” (An) of the section. The “Net Area” (An) is again multiplied by SLF to get the “Effective Net Area” (Ae) of the section.

8. For the design of a single angle for flexure, the parameter LX should be used to specify the value of the term “L” in equations F10-4a, F10-4b, F10-5 and F10-6 of AISC 360-05 and the term “Lb” in equations F10-4, F10-5, F10-6a, and F10-6b of AISC 360-10.

9. When BEAM is 1.0 (default), the design is performed at 13 evenly spaced points along the length of the beam, including start and end points (i.e., 1/12th points or at ends of 12 equal length segments).

![Figure 141: The default sections for design when BEAM 1.0 is used](image)

When BEAM is 0.0, the start and ends along with up to three locations specified in TR.41 Section Specification (on page 2839) are designed.

D1.A.7 Code Checking and Member Selection

Code Checking and Member Selection options are both available in the AISC 360 Unified Code implementation in STAAD.Pro.

For additional information on code checking, refer to:

- D1.B.1.3 Code Checking (on page 1418) for general information and
- TR.49 Code Checking Specification (on page 2852) for input command details

For additional information on member selection refer to:

- D1.B.1.4 Member Selection (on page 1419) for general information and
D1.A.8 Tabulated Results of Steel Design

Results of Code Checking and Member Selection are presented in the output file. The output is clearly marked for the selected specification (AISC 360), edition used (2010 or 2005), and the design method (LRFD or ASD).

The following details are presented on Code Checking of any member:

- Result of Code Checking (Pass / Fail) for the member Number.
- Critical Condition which governed the design and the corresponding Ratio and Location.
- Loads corresponding to the Critical Condition at the Critical Location.
- Section Classification
- Slenderness check report
- Section Capacities in Axial Tension, Axial Compression, Bending, and Shear in both the directions.

If Seismic provisions are to be checked, these follow as described in D1.A.9 Seismic Provision Checking per AISC 341 (on page 1388)

Note: An asterisk following a critical load case number indicates that this load case is a generated load combination. See TR.35 Load Combination Specification (on page 2791) for additional information.

D1.A.9 Seismic Provision Checking per AISC 341

Additional parameters may be specified for member checking or design per AISC 360-05, 360-10, or 360-16 to perform seismic provision checks per the corresponding edition of AISC 341.

The SEISMIC parameter is used to specify seismic provision checks for members. For those members in the AISC 360-05 or 360-10 editions, the MTYP (member type), FRM (frame type), and BRC (bracing type) parameters are used to specify the structure and member details. For the AISC 360-16 edition, only the MTYP (member type) parameter is used to specify the member details. See D1.A.6 Design Parameters (on page 1378) for details on the values used for these parameters.

Provisions Checked

The following checks are performed with the SEISMIC 1 parameter for the AISC 341-05/10 specifications:

- seismic classification of the member - the “worst case” classification is considered among all elements for the member. For
  - For members in an ordinary moment frame (OMF) or intermediate moment frame (IMF), the general section classification of AISC 360-05/10 is used.
  - For members in a special moment frame (SMF), the classification per table I-8-1 of AISC 341-05 or table D1.1 of AISC 341-10, respectively, are used.
- required flexural strength of the member -
  - OMF - $M_r = M_z$ (moment about major axis), per Sec. 11.8 of AISC 341-05 or Sec. E.1.4 of AISC 341-10
  - IMF - $M_r = M_u = R_yZF_y$ (LRFD) or $= R_yZF_y/1.5$ (ASD) per Sec. 10.8 of AISC 341-05 or Sec. D.1.2a of AISC 341-10
  - SMF - $M_r = M_u = R_yZF_y$ (LRFD) or $= R_yZF_y/1.5$ (ASD) per Sec. 9.8 of AISC 341-05 or Sec. D.1.2b of AISC 341-10
- required bracing strength
• Relative bracing for beams - $P_{br} = 0.008M_rC_d/h_0$ per eq. A-6-5 of AISC 360 05/10
• Nodal bracing for beams - $P_{br} = 0.02M_rC_d/h_0$ per eq. A-6-7 of AISC 360 05/10
• Relative bracing for columns - $P_{br} = 0.004P_r$ per eq. A-6-1 of AISC 360 05/10
• Nodal bracing for columns - $P_{br} = 0.01P_r$ per eq. A-6-3 of AISC 360 05/10
• required bracing stiffness: each of the following are multiplied by $1/\phi$ for LRFD or by $\Omega$ for ASD:
  - Relative bracing for beams - $\beta_{br} = 4M_rC_d/(L_b h_0)$ per eq. A-6-6 of AISC 360 05/10
  - Nodal bracing for beams - $\beta_{br} = 10M_rC_d/(L_b h_0)$ per eq. A-6-8 of AISC 360 05/10
  - Relative bracing for columns - $\beta_{br} = 2P_r/L_b$ per eq. A-6-2 of AISC 360 05/10
  - Nodal bracing for columns - $\beta_{br} = 8P_r/L_b$ per eq. A-6-4 of AISC 360 05/10

• required bracing spacing
  - OMF - no requirement
  - IMF - $L_b = 0.17r_yE/F_y$ per Sec 10.8 of AISC 341-05 or Sec. D.1.2a of AISC 341-10
  - SMF - $L_b = 0.086r_yE/F_y$ per Sec 9.8 of AISC 341-05 or Sec. D.1.2b of AISC 341-10

Provisions Checked for AISC 341-16

The following checks are performed with the SEISMIC 1 parameter for the AISC 341-16 specification:

• Section seismic classification against ductility – the section is evaluated to be seismically compact or seismically non-compact for either highly or moderately ductile members (specified using the DUCT parameter)
• Required flexural strength of member checked per D1-1
• Stability bracings of beam
  - Maximum allowed spacing of bracings checked per D1-2
  - Required strength of lateral beam bracing at panel type flange lateral bracing (A-6-5 of AISC 360-16), point type flange lateral bracing (A-6-7 of AISC 360-16), and special flange lateral bracing at plastic hinge locations (D1-4)
  - Required strength of torsional beam bracing at point torsional bracing (A-6-9 of AISC 360-16), continuous torsional bracing, and special torsional bracing at plastic hinge locations (D1-5)
  - Required stiffness of beam bracing at panel type flange lateral bracing and point type flange lateral bracing (A-6-6 of AISC 360-16)
  - Required stiffness of torsional bracing at point torsional bracing (A-6-10 and A-6-11 of AISC 360-16) and continuous torsional bracing (A-6-13 of AISC 360-16)
• Special bracing at plastic hinge locations (D1-6)
• Stability bracings of column
  - Required strength of column bracing at panel type bracing (A-6-1 of AISC 360-16) and point lateral bracing (A-6-3 of AISC 360-16)
• Stability bracing of beam-column
  - Required strength of lateral beam bracing at panel type flange lateral bracing and point type flange lateral bracing (A-6-4 of AISC 360-16)
  - Combination of lateral and torsional bracing
Output

The output for the AISC 341-05/10/16 checks follow the checks for AISC 360-05/10/16 for a member. No seismic provision checks are printed for TRACK 0 output. For TRACK 1 output, the seismic classification and the requirements are given. TRACK 2 adds the details of the classification limits.

Track 1 example for AISC 341-10:

<table>
<thead>
<tr>
<th>SEISMIC PROVISION (AISC 341:10):-</th>
</tr>
</thead>
<tbody>
<tr>
<td>MOMENT FRAME TYPE: IMF</td>
</tr>
<tr>
<td>SEISMIC CLASSIFICATION: FLANGE: Compact WEB: Non-Compact</td>
</tr>
<tr>
<td>REQUIREMENTS:</td>
</tr>
<tr>
<td>EXPECTED FLEXURAL STRENGTH</td>
</tr>
<tr>
<td>BRACING STRENGTH</td>
</tr>
<tr>
<td>BRACING STIFFNESS</td>
</tr>
<tr>
<td>BRACING SPACING</td>
</tr>
</tbody>
</table>

Track 2 example for AISC 341-10:

<table>
<thead>
<tr>
<th>SEISMIC PROVISION (AISC 341:10):-</th>
</tr>
</thead>
<tbody>
<tr>
<td>MOMENT FRAME TYPE: IMF</td>
</tr>
<tr>
<td>SEISMIC CLASSIFICATION: CLASS: l : l md:</td>
</tr>
<tr>
<td>FLANGE: Compact WEB: Compact</td>
</tr>
<tr>
<td>REQUIREMENTS:</td>
</tr>
<tr>
<td>EXPECTED FLEXURAL STRENGTH</td>
</tr>
<tr>
<td>BRACING STRENGTH</td>
</tr>
<tr>
<td>BRACING STIFFNESS</td>
</tr>
<tr>
<td>BRACING SPACING</td>
</tr>
</tbody>
</table>

Track 2 example for AISC 341-16:

<table>
<thead>
<tr>
<th>CHECKS FOR 341-16 SEISMIC CRITERIA</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLEXURAL STRENGTH X</td>
</tr>
<tr>
<td>DEMAND</td>
</tr>
<tr>
<td>-183.8</td>
</tr>
<tr>
<td>Intermediate Results :</td>
</tr>
<tr>
<td>Overstrength factor : Ry = 1.0000</td>
</tr>
<tr>
<td>FLEXURAL STRENGTH Y</td>
</tr>
<tr>
<td>DEMAND</td>
</tr>
<tr>
<td>0.000</td>
</tr>
<tr>
<td>Intermediate Results :</td>
</tr>
<tr>
<td>Overstrength factor : Ry = 1.0000</td>
</tr>
</tbody>
</table>
D1.A.10 AISC 360-16 Design

The checking or selection of members is performed per the following sections of AISC 360-16 and AISC 341-16:

- Section Classification: Chapter B
- Tension Chapter D
- Compression: Chapter E
- Flexure: Chapter F
- Shear: Chapter G
- Torsion (Including checks as per DG-9 for certain profile shapes – see below): Chapter H & Design Guide 9 for Torsion design
- Combined actions – Chapter H for:
  - Axial and/or
  - Flexure and/or
  - Shear and/or
  - Torsion
- Checks for Seismic suitability – AISC 341-16

The following section profile shapes are allowed for design:

- Rolled Shapes
  - I-Shape
  - Channel
  - Single angle
  - Solid Circular bar
  - Solid Rectangular bar
  - Tee
  - HSS rectangular
  - HSS round
- Built Up Shapes
  - Built Up Channel
  - Built Up I
  - Double Channels – back to back
  - Double Channels – face to face
  - Double Angles
  - Built Up Box
  - I With Cover Plates (Top and/or bottom)
- Tapered Shapes
  - I with web tapered
  - Square hollow shape with wall tapered
  - Circular hollow shape diameter tapered
Note: Certain checks (e.g., DG-9 torsion checks) will only be performed for certain profile shapes (non-HSS shapes).

Seismic Design per AISC 341-16
Refer to Provisions Checked for AISC 341-16 (on page 1389) for additional details.

D1.A.10.1 Tension
Design of the members subject to axial tension is performed per chapter D of AISC 360-16.
All the cross-section profiles described in section 1A.10 are considered for tension checks.
Eye bars and pin-connected members are not checked per clauses D5 and D6.

Limit States
The tensile strength is calculated with due consideration to both tensile yielding and tensile rupture effects as per D.2 of the code.
The \( F_y \) and \( F_u \) values would be chosen based on the grade of steel associated with the member, specified by SGR parameter. Note that the implementation also provides a means to create custom steel grades. If SGR is set to 0 ("custom steel"), then the values assigned for FYLD and FU is used respectively.

Slenderness Limits
The code does not mandate the maximum slenderness limit for members in tension. However, the program will check against a maximum slenderness ratio (L/r) which is specified using TMAIN parameter. The length considered to calculate the tension slenderness will be the full member length.

Effective Net Area
The gross area, \( A_g \), and net area, \( A_n \), of tension members shall be determined in accordance with the provisions of Section B4.3 of AISC 360-16.
The effective net area is the net area multiplied by the parameter SLF (shear lag factor).
The net area is the gross area multiplied by the parameter NSF (net section factor).

D1.A.10.2 Compression
Design of the members subject to compression is performed per chapter E of AISC 360-16.
All the cross-section profiles described in section 1A.10 are considered for compression.
The design compressive strength is calculated based on the applicable limit states of flexural buckling, torsional buckling, and flexural-torsional buckling modes as per CLE.1.
The nominal compressive strength, \( P_n \), shall be the lowest value obtained based on the applicable limit states of flexural buckling, torsional buckling, and flexural-torsional buckling. In general, Table.E1.1 is followed for the choice of applicable sections/clauses for the various cross section shapes.

Limit States
Following limit states are checked for flexure per Equations E3-1, E3-2, and E3-3 for without slender elements:
- Flexural buckling about X axis
- Flexural buckling about Y axis
- Flexural buckling about U axis (for unsymmetrical shapes only)
• Flexural buckling about V axis (for unsymmetrical shapes only)
• Flexural-torsional buckling

Flexural-torsional buckling is checked per Equations E4-1, E4-2, and E4-3, E4-4, E4-7. Note the following:
• Check no performed for solid rect or circ bars, thin-walled rect or circ hollow sections, double-channel face-
to-face (only when channels are connected by full weld or tension bolts), or single angles with compact long legs.

The gross area is used for members with non-slender elements whereas the effective area, \( A_n \), is used for slender elements. The effective width of a slender element of any section other than round HSS is calculated for each limit state of compression as per eq. E7-2 or eq. E7-3 depending on the condition mentioned in the specification. The effective area \( (A_n) \) for a slender HSS round section is calculated as per cl. E7.2.

Parameters

The design parameters that affect compression are:
• LEG
• LZ
• LY
• LX
• KZ
• KY
• KZ
• SNU
• CSP

Slenderness Limits

The code does not mandate the maximum slenderness limit for members in compression. However, the program will check against a maximum slenderness ratio \((L/r)\) which is specified using MAIN parameter whose default value is 180. The length considered to calculate the compression slenderness will be the full member length.

**D1.A.10.3 Flexure**

Design strength in flexure is calculated as per clause F1. The

Limit States

The nominal flexural strength is calculated for different limit states as per sections F2 through F13.

The following limit states are checked for flexure:
• Flexural Yielding about X and Y axis
• Flexural Yielding about U and V axis (for single angle only)
• Lateral-torsional Buckling about X and Y axis
• Lateral-torsional Buckling about U axis (for single angle only)
• Flange Local Buckling
• Web Local Buckling
• Wall Local Buckling (for HSS Round only)
• Compression Flange Yielding
• Tension Flange Yielding

Parameters
The design parameters that affect flexure are:

- UNT
- UNB
- UNL
- UNR
- FLX
- SNU
- CB

**D1.A.10.4 Shear**

Design of the members subject to shear is performed per chapter G of AISC 360-16. All the cross-section profiles described in section 1A.10 are considered for shear. An option to consider tension field action is available. Also, the option to consider the shear capacity of I sections or channel shapes with transverse web stiffeners.

The program does not consider any specific checks for web openings in clause G7.

The design of transverse web stiffeners is not performed.

**Limit States**

The design shear strength is calculated per G1.

The nominal shear strength is calculated for the following sections:

- I-Shapes and Channels G2
- Single Angles and Tees G3
- Rect HSS, Box Sections, and other Doubly Symmetric Members G4
- Round HSS G5
- Shear capacity along weak axis in Doubly Symmetric and Singly Symmetric Shapes G6

**D1.A.10.5 Combined Forces and Torsion**

The effects of combined actions of axial force with major and minor axis moments are accounted for the following:

Doubly and Singly symmetric members subject to flexure and compression - H1.1

- Rolled I
- Rolled Channel
- Solid circular bar
- Solid rectangular bar
- Tee
- HSS Rectangle
- HSS Round
- Built Up Channel
- Built Up I
- Double Channels – back to back
- Double Angles
- Built Up Box
• I Profile With Cover Plates (Top and/or bottom)

Doubly and Singly symmetric members subject to flexure and tension - H1.2

• Rolled I
• Rolled Channel
• Solid circular bar
• Solid rectangular bar
• Tee
• HSS Rectangle
• HSS Round
• Built Up Channel
• Built Up I
• Double Channels – back to back
• Double Angles
• Built Up Box
• I Profile With Cover Plates (Top and/or bottom)

Doubly symmetric rolled compact members subject to single axis flexure and compression – H1.3

If you prefer to use only cl. H1.1 even when cl. H1.3 is applicable, this may be done with the INT parameter.

Unsymmetrical and other members subject to Flexure and Axial Force - H2

This section is used to check single angles.

Members Subject to Torsion and combined Torsion, Flexure, Shear, and Axial Force - H3

Checks for members subject to pure torsion or subject to torsion combined with other effects such as flexure or axial loads will be performed based on the profile shape of the member. Only the following section profiles are considered when performing torsion checks:

• Closed profiles
  • HSS Rectangle
  • HSS Round
  • Solid Circular bar
  • Solid Rectangular bar

• Open profiles
  • Rolled I
  • Rolled Channel
  • Tee
  • Single angle
  • Built-up I
  • Built-up Channel (Only if symmetric about X-X)
  • I Profile With Cover Plates (Top and/or bottom)

The implementation will perform the AISC 360-16 code checks as per clauses H.3.1 and H.3.2 for circular & rectangular hollow shapes. For all other shapes listed above, the component will follow the design methods as per AISC Design Guide -9: Torsional Analysis of Structural Steel Members (DG-9).
For non-HSS members subject to torsion and combined stress, the program cannot detect the load-end restraint condition automatically from the model. You can specify this through TND parameter. Also, for some cases the distance where the torque is applied must be provided through the ALH parameter.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
- RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10 (on page 94)

**D1.A.10.6 Updates to AISC 360-16**

This section highlights the changes implemented from AISC 360-16 from AISC 360-10.

Section Classification

The width to thickness ratio limits for members in flexure have been updated for the following element types as per AISC 360-16:

- Stems of tees (14) are considered slender when \( \frac{d}{t} > 1.52 \frac{E}{F_y} \).
- Flanges of box sections (21) are considered slender when \( \frac{b}{t} > 1.49 \frac{E}{F_y} \).

**CHAPTER E Compression Checks: Torsional & Flexural-Torsional Buckling Calculations**

Design for the following profile shapes/conditions have been updated to use the corresponding clauses/equations from AISC 360-16:

2.a Cl. E4: Non-Slender Double Angles & Tee Shapes
- Calculation of \( F_{ez} \) and \( F_{ey} \) to use (E4-9) and (E4-8).
- Calculation of \( F_{e} \) to use (E4-3).
- Calculation of \( F_{cr} \) to use (E3-2) or (E3-3).
- Calculation of \( P_{n} \) to use (E4-1).

2.b Cl. E5: Single-Angle Compression Members
- Flexural torsional buckling limit for single angles changed to \( \frac{b}{t} \leq 0.71 \frac{E}{F_y} \).

2.c Cl. E7: Compression Members with Slender Elements
- Calculation of nominal compressive strength and critical stress \( F_{cr} \) updated to follow the updated criteria from Clause E7.

**CHAPTER F Flexure Checks**

Design for flexural members has been updated to use the following:

3.a Cl.F4 : Non-Standard[other] I Shaped Members

Lateral Torsional Buckling:
The calculation of effective radius of gyration, \( r_t \), updated to use the AISC 360-16 equation F4-11 for I-shapes with a rectangular compression flange:

\[
r_t = \frac{b_f c}{\sqrt{12(1 + \frac{1}{6a_w})}} \quad \text{(F4-11)}
\]

3.b Cl.A : Tension Flange Yielding
The tension flange yielding checks have been updated to use the updated web plastification factor calculations as per equations F4-16a to F4-17.

1. When \( \frac{I_{yc}}{I_y} > 0.23 \)
   
   i. When \( \frac{h_c}{t_w} \leq \lambda_{pw} \)
   
   \[ R_{pt} = \frac{M_p}{M_{yt}} \]  
   
   \[(F4-16a)\]

   ii. When \( \frac{h_c}{t_w} > \lambda_{pw} \)
   
   \[ R_{pt} = \left( \frac{M_p}{M_{yt}} - \left( \frac{M_p}{M_{yt}} - 1 \right) \left( \frac{\lambda - \lambda_{pw}}{\lambda_{re} - \lambda_{pw}} \right) \right) \leq \frac{M_p}{M_{yt}} \]  
   
   \[(F4-16b)\]

2. When \( \frac{I_{yc}}{I_y} \leq 0.23 \)
   
   \[ R_{pt} = 1.0 \]  
   
   \[(F4-17)\]

3. c CLF7: Flexural checks for Square and Rectangular HSS & Box Sections

   - **CLF7.2**: Flange local buckling checks updated to use F7-4 & F7-5 of AISC 360-16 to calculate the effective width of slender elements.
     
     - For HSS
       
       \[ b_c = 1.92 t_f \sqrt{\frac{E}{F_y}} \left( 1 - \frac{0.38}{b/t_f} \sqrt{\frac{E}{F_y}} \right) \leq b \]  
       
       \[(F7-4)\]
     
     - For box sections
       
       \[ b_c = 1.92 t_f \sqrt{\frac{E}{F_y}} \left( 1 - \frac{0.34}{b/t_f} \sqrt{\frac{E}{F_y}} \right) \leq b \]  
       
       \[(F7-5)\]
   
   - **CLF7.3**: Web local buckling checks updated to use F7-6 through to F7-9 of AISC 360-16.
     
     - For sections with noncompact webs
       
       \[ M_n = M_p - (M_p - F_y S) \left( 0.305 \frac{b_c}{t_w} \sqrt{\frac{E}{F_y}} - 0.738 \right) \leq M_p \]  
       
       \[(F7-6)\]
       
       where
       
       \( h \) = depth of web, as defined in Section B4.1b, in. (mm)
     
     - For sections with slender webs
       
       - compression flange yielding
         
         \[ M_n = R_{pg} F_y S \]  
         
         \[(F7-7)\]
       
       - compression flange local buckling
         
         \[ M_n = R_{pg} F_{cr} S_{xc} \]  
         
         \[(F7-8)\]
       
       and
       
       \[ F_{cr} = \frac{0.9 E k_c}{\left( \frac{b}{t_f} \right)^2} \]  
       
       \[(F7-9)\]
       
       where
       
       \( R_{pg} \) = defined by Equation F5-6 with \( a_w = 2h t_w / (bt_f) \)
\[ k_c = 4.0 \]

- CL.F7.4: Lateral torsional buckling checks updated to use F7-6 through to F7-13 of AISC 360-16.

3.d CL.F9: Flexural checks for Tees and Double Angles

- Yielding checks for Tee sections updated to use Eqn F9-4 of AISC 360-16.
- Yielding checks for Double Angle sections updated to use Eqn F9-5 of AISC 360-16.

\[ M_p = 1.5M_y \] (F9-5)

- Lateral torsional buckling checks updated to use Eqns F9-6 through to F9-13 of AISC 360-16
- Flange Local Buckling calculations of Tees and double angle legs updated to include criteria as per CL. F9.3 (b) of AISC 360-16
- Web local buckling checks for Stem of Tee sections updated to use Cl.F9.4 - Eqns F9-17 through to F9-19 of AISC 360-16.
- Web local buckling checks for legs of double angle sections updated to use Cl.F9.4 (b) of AISC 360-16.

3.e CL.F10: Flexural checks for Single Angles

- Lateral Torsional Buckling checks updated to use Eqns F10-4 through to F10-5b of AISC 360-16.

CHAPTER G Shear Checks

4.a CL.G2: I Shapes and Channels

- Calculation of Shear strength coefficient Cv updated to use Eqns G2-2 to G2-4 of AISC 360-16
- Calculation of Shear buckling coefficient kv updated to use Eqns from section G2 of AISC 360-16.
- Criteria for consideration of Tension field action updated as per Section G2.2 of AISC 360-16.
- Shear strength calculations for cases with tension field action updated to use Eqns G2-6 through to G2-11 of AISC 360-16.

D1.A.10.7 Output

Preceding the member output will be a series of design statements regarding axis conventions, nomenclature, notes, and abbreviations.

TRACK 0 Output

This format reports the design summary for a member indicating the pass/fail status, the critical design ratio, the most critical load case ID, and the code clause that produced the critical design ratio.
**TRACK 1 Output**

This format produces a more detailed report for the member. In addition to the items from a TRACK 0 output, this will report a summary of the checks for each of the individual effects considered during design. This will also provide the section properties and design criteria that were used for the design process. A summary of AISC341-16 checks, if performed, will also be included with the TRACK 1 output. Refer to [Provisions Checked for AISC 341-16](on page 1389) for additional details.

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile: ST W18X50</th>
<th>Status:</th>
<th>Ratio:</th>
<th>Loadcase:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(AISC SECTIONS)</td>
<td>PASS</td>
<td>0.703</td>
<td>3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Location:</th>
<th>Ref:</th>
<th>Pz:</th>
<th>T</th>
<th>Vy:</th>
<th>Vx:</th>
<th>Tz:</th>
<th>My:</th>
<th>Mx:</th>
</tr>
</thead>
<tbody>
<tr>
<td>17.50</td>
<td>Cl.F2.1</td>
<td>0.000</td>
<td>T</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td></td>
<td>-266.4</td>
</tr>
</tbody>
</table>

**SLENDERNESS**

Actual Slenderness Ratio : 254.294

Allowable Slenderness Ratio : 300.000

**STRENGTH CHECKS**

Critical L/C : 3

Ratio : 0.703(PASS)

Loc : 17.50
Condition : Cl.F2.1

**SECTION PROPERTIES** (LOC: 17.50, PROPERTIES UNIT: IN )

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.470E+01</td>
<td>8.550E+00</td>
<td>6.390E+00</td>
<td>8.000E+02</td>
<td>4.010E+01</td>
<td>1.240E+00</td>
<td>8.889E+01</td>
<td>8.889E+01</td>
<td>1.010E+02</td>
<td>1.069E+01</td>
<td>1.069E+01</td>
<td>1.660E+01</td>
</tr>
</tbody>
</table>
Design

D. Design Codes

---

\[ Cw : 3.046E+03 \quad x0 : 0.000E+00 \quad y0 : 0.000E+00 \]

---

**MATERIAL PROPERTIES**

\[ \text{Fyld: 7200.000} \quad \text{Fu: 9359.999} \]

---

**Actual Member Length:** 35.000

**Design Parameters**

\[ Kx: 1.00 \quad Ky: 1.00 \quad NSF: 1.00 \quad SLF: 1.00 \quad CSP: 1.00 \]

---

**COMPRESSION CLASSIFICATION (L/C: 3 LOC: 420.00)**

<table>
<thead>
<tr>
<th>Flange</th>
<th>l</th>
<th>lp</th>
<th>lr</th>
<th>CASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>NonSlender</td>
<td>6.58</td>
<td>N/A</td>
<td>13.49</td>
<td>Table.4.1a.Case1</td>
</tr>
<tr>
<td>Slender</td>
<td>47.49</td>
<td>N/A</td>
<td>35.88</td>
<td>Table.4.1a.Case5</td>
</tr>
</tbody>
</table>

**FLEXURE CLASSIFICATION** (L/C: 3 LOC: 420.00)

<table>
<thead>
<tr>
<th>Flange</th>
<th>l</th>
<th>lp</th>
<th>lr</th>
<th>CASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compact</td>
<td>6.58</td>
<td>9.15</td>
<td>24.08</td>
<td>Table.4.1b.Case10</td>
</tr>
<tr>
<td>Compact</td>
<td>47.49</td>
<td>90.55</td>
<td>137.27</td>
<td>Table.4.1b.Case15</td>
</tr>
</tbody>
</table>

---

**TRACK 2 Output**

This format expands on the TRACK 1 format and will provide the final capacities along with the primary intermediate values that were considered during the design for each effect considered. This will also provide the section properties and design criteria that were used for the design process.

---

**STAAD SPACE**

NO. 4

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)**

*ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).*

**Member:** 1

**Member No:** 1 **Profile:** ST W18X50 (AISC SECTIONS)

---

*STAAD.Pro 1400 User Manual*
<table>
<thead>
<tr>
<th>Status:</th>
<th>PASS</th>
<th>Ratio:</th>
<th>0.703</th>
<th>Loadcase:</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>17.50</td>
<td>Ref:</td>
<td>Cl.F2.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>0.000</td>
<td>T</td>
<td>Vy:</td>
<td>0.000</td>
<td>Vx:</td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My:</td>
<td>0.000</td>
<td>Mx:</td>
<td>-266.4</td>
</tr>
</tbody>
</table>

---

**SLENDERNESS**

Actual Slenderness Ratio : 254.294
Allowable Slenderness Ratio : 300.000  LOC : 0.00

---

**STRENGTH CHECKS**

Critical L/C : 3  Ratio : 0.703(PASS)
Loc : 17.50  Condition : Cl.F2.1

---

**SECTION PROPERTIES (LOC: 17.50, PROPERTIES UNIT: IN )**

<table>
<thead>
<tr>
<th>Ag</th>
<th>1.470E+01</th>
<th>Axx</th>
<th>8.550E+00</th>
<th>Ayy</th>
<th>6.390E+00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ixx</td>
<td>8.000E+02</td>
<td>Iyy</td>
<td>4.010E+01</td>
<td>J</td>
<td>1.240E+00</td>
</tr>
<tr>
<td>Sxx+</td>
<td>8.889E+01</td>
<td>Sxx-</td>
<td>8.889E+01</td>
<td>Zxx</td>
<td>1.010E+02</td>
</tr>
<tr>
<td>Syy+</td>
<td>1.069E+01</td>
<td>Syy-</td>
<td>1.069E+01</td>
<td>Zyy</td>
<td>1.660E+01</td>
</tr>
<tr>
<td>Cw</td>
<td>3.046E+03</td>
<td>x0</td>
<td>0.000E+00</td>
<td>y0</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

---

**MATERIAL PROPERTIES**

<table>
<thead>
<tr>
<th>Fyld</th>
<th>7200.000</th>
<th>Fu</th>
<th>9359.999</th>
</tr>
</thead>
</table>

---

Actual Member Length: 35.000

---

Design Parameters

<table>
<thead>
<tr>
<th>Kx</th>
<th>1.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ky</td>
<td>1.00</td>
</tr>
<tr>
<td>NSF</td>
<td>1.00</td>
</tr>
<tr>
<td>SLF</td>
<td>1.00</td>
</tr>
<tr>
<td>CSP</td>
<td>1.00</td>
</tr>
</tbody>
</table>

---

**COMPRESSION CLASSIFICATION (L/C: 3 LOC: 420.00)**

1  lp  lr  CASE
Flange: NonSlender  6.58  N/A  13.49  Table.4.1a.Case1
Web : Slender  47.49  N/A  35.88  Table.4.1a.Case5

FLEXURE CLASSIFICATION  (L/C:  3 LOC:  420.00)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th>CASE</th>
</tr>
</thead>
</table>
| Flange: Compact  6.58  9.15  24.08  Table.4.1b.Case10
Web : Compact  47.49  90.55  137.27  Table.4.1b.Case15

---

STAAD SPACE  -- PAGE NO.  5

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :  1 Contd.
---

CHECKS FOR AXIAL TENSION

TENSILE YIELDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>661.5</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results :
Nom. Ten. Yld Cap : Pn = 735.00 kip  Eq.D2-1

TENSILE RUPTURE

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>716.6</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results :
Effective area : Ae = 0.10208 ft2  Eq.D3-1
CHECKS FOR AXIAL COMPRESSION

FLEXURAL BUCKLING X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>498.5</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Effective Slenderness : \( L_{cx}/r_{x} = 56.933 \) Cl.E2

Elastic Buckling Stress : \( F_{ex} = 12716 \) kip/ft² Eq.E3-4

Crit. Buckling Stress : \( F_{crx} = 5680.7 \) kip/ft² Eq.E3-2

Nom. Flexural Buckling : \( P_{nx} = 553.88 \) kip Eq.E7-1

FLEXURAL BUCKLING Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>51.36</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Effective Slenderness : \( L_{cy}/r_{y} = 254.29 \) Cl.E2

Elastic Buckling Stress : \( F_{ey} = 637.37 \) kip/ft² Eq.E3-4

Crit. Buckling Stress : \( F_{cry} = 558.97 \) kip/ft² Eq.E3-3

Nom. Flexural Buckling : \( P_{ny} = 57.061 \) kip Eq.E7-1

FLEX-TOR-BUCKLING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>259.3</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
| Elastic F-T-B Stress      : Fe     =  3217.8 kip/ft²    Eq.E4-2 |
|---------------------------|-----------------|-----------------|
| Crit. F-T-B Stress        : Fcr    =  2822.3 kip/ft²    Eq.E3-2 |
| Nom. Flex-tor Buckling    : Pn     =  288.11 kip        Eq.E7-1 |

STAAD SPACE
-- PAGE NO. 6

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 1 Contd.

CHECKS FOR SHEAR

SHEAR ALONG X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>230.8</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Coefficient Cv Along X : Cv     =  1.0000    Eq.G2-9
Coefficient Kv Along X : Kv     =  1.2000    Cl.G6
Nom. Shear Along X     : Vnx     =  256.50 kip    Eq.G6-1

SHEAR ALONG Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>30.45</td>
<td>191.7</td>
<td>0.159</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Coefficient Cv Along Y : Cv     =  1.0000    -
Coefficient Kv Along Y : Kv     =  5.3400 kip    Eq.G2-5
Nom. Shear Along Y     : Vny     =  191.70 kip    Eq.G2-1
### Checks for Bending

**Flex Yielding (X)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>-266.4</td>
<td>378.8</td>
<td>0.703</td>
<td>Cl.F2.1</td>
<td>3</td>
<td>17.50</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom Flex Yielding Along X : Mnx = 420.83 kip-ft  
Eq. F2-1

**Flex Yielding (Y)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>62.25</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom Flex Yielding Along Y : Mny = 69.167 kip-ft  
Eq. F6-1

---

### Checks for Axial Bend Interaction

---
D1.B. American Codes - Steel Design per AISC 9th Edition

D1.B.1 Working Stress Design

D1.B.1.1 Allowables per AISC Code

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by the American Institute of Steel Construction (AISC) Code. The ninth edition of the AISC Code, as published in 1989, is used as the basis of this design (except for tension stress). Because of the size and complexity of the AISC codes, it would not be practical to describe every aspect of the steel design in this manual. Instead, a brief description of some of the major allowable stresses are described herein.

D1.B.1.1.1 Tension Stress
Allowable tensile stress on the net section is calculated as:

\[ F_t = 0.60 \cdot F_y \]

D1.B.1.1.2 Shear Stress
Allowable shear stress on the gross section,

\[ F_v = 0.4 \cdot F_y \]

D1.B.1.1.3 Stress Due To Compression
Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the formula E-1 in the AISC Code, when the largest effective slenderness ratio \((Kl/r)\) is less than \(C_c\). If \(Kl/r\) exceeds \(C_c\), allowable compressive stress is decreased as per formula 1E2-2 of the Code.

\[ C_c = \sqrt{\frac{2n^2E}{F_y}} \]

D1.B.1.1.4 Bending Stress
Allowable bending stress for tension and compression for a symmetrical member loaded in the plane of its minor axis, as given in Section 1.5.1.4 is:
\[ F_b = 0.66F_y \]

If meeting the requirements of this section of:

a. \[ b_f/2t_f \leq 65/\sqrt{(F_y)} \]

b. \[ b_f/t_f \leq 190/\sqrt{(F_y)} \]

c. \[ d/t \leq 640(1 - 3.74(f_a/F_y))/\sqrt{(F_y)} \] when \( f_a/F_y < 0.16 \), or than 257/\sqrt{(F_y)} if \( f_a/F_y > 0.16 \)

d. The laterally unsupported length shall not exceed 76.0 \( b_f/F_y \) (except for pipes or tubes), nor 20,000/(d \( F_y/A_f \))

e. The diameter:thickness ratio of pipes shall not exceed 3,300/\sqrt{(F_y)}

If for these symmetrical members, \( b_f/2t_f \) exceeds 65/\sqrt{(F_y)} but is less than 95/\sqrt{(F_y)}, \[ F_b = F_y(0.79 - 0.002(b_f/2t_f)\sqrt{(F_y)}) \]

For other symmetrical members which do not meet the above, \( F_b \) is calculated as the larger value computed as per AISC formulas F1-6 or F1-7 and F1-8 as applicable, but not more than 0.60\( F_y \). An unstiffened member subject to axial compression or compression due to bending is considered fully effective when the width-thickness ratio is not greater than the following:

- 76.0/\sqrt{(F_y)} for single angles or double angles with separators
- 95.0/\sqrt{(F_y)} for double angles in contact
- 127.0/\sqrt{(F_y)} for stems of tees

When the actual width-thickness ratio exceeds these values, the allowable stress is governed by BS of the AISC code.

Tension and compression for the double symmetric (I & H) sections with \( b_f/2t_f \) less than 65/\sqrt{(F_y)} and bent about their minor axis, \( F_b = 0.75 F_y \). If \( b_f/2t_f \) exceeds 65/\sqrt{(F_y)} but is less than 95/\sqrt{(F_y)}, \[ F_b = F_y(1.075 - 0.005(b_f/2t_f)\sqrt{(F_y)}) \]

For tubes, meeting the subparagraphs b and c of this Section, bent about the minor axis, \( F_b = 0.66F_y \); failing the subparagraphs B and C but with a width:thickness ration less than 238/\sqrt{(F_y)}, \[ F_b = 0.6F_y \].

D1.B.1.1.5 Combined Compression and Bending

Members subjected to both axial compression and bending stresses are proportioned to satisfy AISC formula H1-1 and H1-2 when \( f_a/F_a \) is greater than 0.15, otherwise formula H1-3 is used. It should be noted that during code checking or member selection, if \( f_a/F_a \) exceeds unity, the program does not compute the second and third part of the formula H1-1, because this would result in a misleadingly liberal ratio. The value of the coefficient \( C_m \) is taken as 0.85 for sidesway and \( 0.6 - 0.4(M1/M2) \), but not less than 0.4 for no sidesway.

D1.B.1.1.6 Singly Symmetric Sections

For double angles and Tees which have only one axis of symmetry, the \( KL/r \) ratio about the local Y-Y axis is determined using the clauses specified on page 3-53 of the AISC ASD 9th ed. Manual.

D1.B.1.1.7 Torsion per Publication T114

The AISC 89 code of specifications for steel design currently does not have any provisions specifically meant for design of sections for Torsion. However, AISC has published a separate document called *Torsional Analysis of Steel Members* which provides guidelines on transforming torsional moments into normal stresses and shear stresses which can then be incorporated into the interaction equations explained in Chapter H of the AISC 89 code. The guidelines of the publication have now been incorporated into the AISC-89 steel design modules of STAAD.

To consider stresses due to torsion in the code checking or member selection procedure, specify the parameter TORSION with a value of 1.0. See D1.B.1.2 Design Parameters (on page 1409) for more details.

Methodology
If the user were to request design for torsion, the torsional properties required for calculating the warping normal stresses, warping shear stresses and pure shear stresses are first determined. These depend on the "boundary" conditions that prevail at the ends of the member. These boundary conditions are defined as "Free", "Pinned" or "Fixed". They are explained below:

**Free** represents the boundary condition such as that which exists at the free end of a cantilever beam. It means that there is no other member connected to the beam at that point.

**Pinned** represents the condition that corresponds to either a pinned support defined at the joint through the Support command or a release of any of the moments at the joint through a Member Release Specification.

**Fixed** represents the condition where a fixed support exists at the joint. In the absence of a support at that joint, it represents a condition where a rigid frame connection exists between the given member and at least one other member connected to that joint. Also, no member releases should be present at that joint on the given member.

After the boundary conditions are determined, the normal and shear stresses are determined. The guidelines specified in the publication T114 for concentrated torsional moments acting at the ends of the member are used to determine these stresses.

The warping normal stresses are added to the axial stresses caused by axial load. These are then substituted into the interaction equations in Chapter H of the AISC 89 code for determining the ratio. The plane shear and warping shear stresses are added to the shear stresses caused by actual shear forces and compared against the allowable shear stresses on the cross section.

**Torsional boundary conditions at a joint where a FIXED BUT type of support is specified**

If the end of a the member is declared a FIXED BUT type of support, the torsional boundary conditions at that end are determined in the following manner.

a. If all of the 3 translational degrees of freedom at that support are either free to displace, or have a spring, then, that end of the member is considered torsionally FREE.

    Example:
    45 FIXED BUT MX MY MZ KFX 75 KFY 115
    In this example, at joint 45, a spring has been specified along KFX and KFY, and, no restraint is provided for translation along global Z. So, the member which has joint 45 as one of its nodes is considered torsionally free at joint 45.

b. If any of the 3 translational degrees of freedom at that support are restrained, and, any of the moment degrees of freedom are unrestrained or have a spring, then, that end of the member is considered torsionally PINNED.

    Examples:
    78 FIXED BUT FX MZ
    In this example, joint 78 is prevented from translation along global Y and Z, and free to rotate about global Z. So, the member which has joint 78 as one of its nodes is considered torsionally PINNED at joint 78.

    17 FIXED BUT MX MY
    In this example, joint 17 is prevented from translation along global X, Y and Z, and free to rotate about global X and Y. So, the member which has joint 17 as one of its nodes is considered torsionally PINNED at joint 17.

    85 FIXED BUT FZ MZ KFY 1.0E8 KMX 1.6E6
In this example, the joint is prevented from translation along global X, has a rotational spring for resisting moments about global X and is free to rotate about global Z. So, the member which has joint 85 as one of its nodes is considered torsionally PINNED at joint 85.

Restrictions

This facility is currently available for Wide Flange shapes (W, M & S), Channels, Tee shapes, Pipes and Tubes. It is not available for Single Angles, Double Angles, members with the PRISMATIC property specification, Composite sections (Wide Flanges with concrete slabs or plates on top), or Double Channels. Also, the stresses are calculated based on the rules for concentrated torsional moments acting at the ends of the member.

D1.B.1.1.8 Design of Web Tapered Sections

Appendix F of AISC-89 provides specifications for design of Web-Tapered members. These specifications have been incorporated into STAAD to perform code checking on web tapered wide flange shapes. Please note that member selection cannot be performed on web-tapered members.

D1.B.1.1.9 Slender Compression Elements

For cross sections with elements which fall in the category of slender as per Table B5.1 of the AISC ASD code (the others being compact and non-compact), the rules of Appendix B of the code have been implemented. For stiffened compression elements, the effective cross section properties are calculated and used. For unstiffened compression elements, the allowable stresses are reduced per the Appendix.

D1.B.1.2 Design Parameters

The program contains a large number of parameter names which are needed to perform designing and code checking. These parameter names, with their default values, are listed in Table 2-3. These parameters communicate design decisions from the engineer to the program.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ (k value in local z-axis) value of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instructions. Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

The parameters PROFILE, DMAX, and DMIN are only used for member selection.

Table 91: AISC (9th Ed.) Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AXIS</td>
<td>1</td>
<td>Select axis about which single angles are design</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Design single angles for bending about their principle axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = Design single angles for bending about their geometric axis.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| BEAM           | 1.0           | Used to specify the number of sections at which the member design is evaluated.  
0.0 = design at start and end nodes and those locations specified by the SECTION command.  
1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes.  
**Note:** See [D1.A.6 Design Parameters](on page 1378). |
| BMAX           | 83.3333 ft    | Maximum allowable width of the flange. Used in the design of tapered sections. |
| CAN            | 0             | Specifies the method used for deflection checks  
0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2.  
1 = deflection check based on the principle that maximum deflection is of the cantilever type (see note 1) |
<p>| CB             | 1.0           | Cb value as used in Section 1.5 of AISC. Use 0.0 to direct the program to calculated Cb. Any other value be used in lieu of the program calculated value. |
| CDIA           | 0.0           | The diameter of circular openings. If a member has more than one circular opening, they can have different diameters. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHOLE</td>
<td>NONE</td>
<td>Section locations of circular openings along the length of the member. Maximum three locations can be specified for each member when there is no rectangular opening.</td>
</tr>
<tr>
<td>CMP</td>
<td>0</td>
<td>Composite action with connectors</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 = design as non-composite beam</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = design as a composite beam if the slab is in bending compression throughout the span, design as a non-composite beam if the slab is in tension anywhere along the span</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = design as a composite beam only. Ignore moments which cause tension in the slab.</td>
</tr>
<tr>
<td>CMZ</td>
<td>0.85 for sidesway and calculated for no sidesway</td>
<td>Cm value in local y and z axes, respectively.</td>
</tr>
<tr>
<td>CYC</td>
<td>500,000</td>
<td>Cycles of maximum stress to which the shear connectors are subject.</td>
</tr>
<tr>
<td>DFF</td>
<td>none (mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DIA</td>
<td>0.625 in.</td>
<td>Diameter of the shear connectors</td>
</tr>
<tr>
<td>DINC</td>
<td>1 in</td>
<td>Incremental depth value used in the design of tapered sections.</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot; (see note 1)</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot; (see note 1)</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000 in.</td>
<td>Maximum allowable section depth.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in.</td>
<td>Minimum allowable section depth.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DR1</td>
<td>0.4</td>
<td>Ratio of moment due to dead load applied before concrete hardens to total moment.</td>
</tr>
<tr>
<td>DR2</td>
<td>0.4</td>
<td>Ratio of moment due to dead load applied after concrete hardens to total moment.</td>
</tr>
<tr>
<td>ELECTRODE</td>
<td>1</td>
<td>Weld material to be used for reinforced opening.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 = E60XX</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = E70XX</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = E80XX</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 = E90XX</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4 = E100XX</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5 = E110XX</td>
</tr>
<tr>
<td>FBINC</td>
<td>0</td>
<td>Incremental bottom flange width used in the design of tapered sections. In this case, the top flange width will remain unchanged.</td>
</tr>
<tr>
<td>FLX</td>
<td>1</td>
<td>Single angle member bracing</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Single angle member is not fully braced against lateral torsional buckling.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = Single angle member is fully braced against lateral torsional buckling.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 = Single angle member is braced against lateral torsional buckling at the point of maximum moment.</td>
</tr>
<tr>
<td>FPC</td>
<td>3.0 ksi</td>
<td>Compressive strength of concrete at 28 days</td>
</tr>
<tr>
<td>FSS</td>
<td>1</td>
<td>Is the full section to be used for shear design?</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 = No (False)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Yes (True)</td>
</tr>
<tr>
<td>FTBINC</td>
<td>0</td>
<td>Incremental flange width (top and bottom) used in the design of tapered sections.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>FTINC</td>
<td>0</td>
<td>Incremental top flange width used in the design of tapered sections. In this case, the bottom flange width will remain unchanged.</td>
</tr>
<tr>
<td>FU</td>
<td>Depends on FYLD</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If FYLD &lt; 40 KSI, then FU = 58 KSI</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If 40 KSI ≤ FYLD ≤ 50 KSI, then FU = 60 KSI</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If FYLD &gt; 50 KSI, then FU = 65 KSI</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 KSI</td>
<td>Yield strength of steel in current units.</td>
</tr>
<tr>
<td>HECC</td>
<td>0.0</td>
<td>Eccentricity of opening with respect to the centerline of the member.</td>
</tr>
<tr>
<td>KX</td>
<td>1.0</td>
<td>K value used in computing KL/r for flexural torsional buckling for tees and double angles.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis.</td>
</tr>
<tr>
<td>LX</td>
<td>Member Length</td>
<td>Length value used in computing KL/r for flexural torsional buckling for tees and double angles.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length used to calculate slenderness ratio for buckling about the local y-axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as LY, but in the local z-axis.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>MAIN</td>
<td>0.0</td>
<td>Toggles the slenderness check  0.0 = check for slenderness  1.0 = suppress slenderness check  Any value greater than 1 = Allowable KL/r in compression.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>OVR</td>
<td>1.0</td>
<td>Overstress factor. All the allowable stress are multiplied by this number. It may be assigned any value greater than 0.0. It is used to communicate increases in allowable stress for loads like wind and earthquake.</td>
</tr>
<tr>
<td>PLTHICK</td>
<td>0.0</td>
<td>Thickness of cover plate welded to the bottom flange of the composite beam.</td>
</tr>
<tr>
<td>PLWIDTH</td>
<td>0.0</td>
<td>Width of cover plate welded to the bottom flange of the composite beam.</td>
</tr>
<tr>
<td>PROFILE</td>
<td></td>
<td>Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual to allowable stress.</td>
</tr>
<tr>
<td>RDIM</td>
<td>0.0</td>
<td>Dimensions of rectangular openings (at each section, RDIM has a length term and a depth term – see syntax below). If a member has more than one rectangular opening they can have different dimensions.</td>
</tr>
<tr>
<td>RHOLE</td>
<td>None</td>
<td>Section locations of rectangular openings along the length of the member. Maximum three locations can be specified for each member when there is no circular opening.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>RBHEIGHT</td>
<td>0.0</td>
<td>Height of ribs in the form steel deck.</td>
</tr>
<tr>
<td>RBWIDTH</td>
<td>2.5 in.</td>
<td>Width of ribs in the form steel deck.</td>
</tr>
</tbody>
</table>
| SHE            | 0             | Option for calculating actual shear stress.  
|                |               | 0 = Compute the shear stress using VO/Ib  
|                |               | 1 = Computer the shear stress based on the area of the section element. |
| SHR            | 0             | Indicates use of temporary shoring during construction.  
|                |               | 0 = Without shoring  
|                |               | 1 = With shoring |
| SSY            | 0.0           | Sidesway  
|                |               | 0.0 = Sidesway in local y-axis.  
|                |               | 1.0 = No sidesway |
| SSZ            | 0.0           | Same as SSY, but in local z-axis. |
| STIFF          | Member Length or depth of beam, whichever is greater | Spacing of stiffeners for plate girder design. |
| STP            | 1             | Section type as defined in ASD Manual table.  
|                |               | 1 = Rolled  
|                |               | 2 = Welded |
| TAPER          | 1.0           | Design basis for tapered members  
|                |               | 0.0 = Design tapered I-section based on rules of Chapter F and Appendix B of AISC only. Due not use the rules in Appendix F of AISC-89.  
<p>|                |               | 1.0 = Design tapered I-sections based on the rules of Appendix F of AISC-89. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>THK</td>
<td>4.0 in.</td>
<td>Thickness of concrete slab or the thickness of concrete slab above the form steel deck.</td>
</tr>
<tr>
<td>TMAIN</td>
<td>300</td>
<td>Any value greater than 1 = Allowable KL/r in tension.</td>
</tr>
</tbody>
</table>
| TORSION        | 0.0           | Toggles the check for torsion  
0.0 = No torsion check is performed  
1.0 = Perform torsion check based on rules of AISC T114. |
| TRACK          | 0.0           | Controls the level of detail to which results are reported:  
0 = minimum detail  
1 = intermediate detail  
2 = maximum detail (see Figure 2.1) |
| UNB            | Member Length | Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the bottom flange. |
| UNT            | Member Length | Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the top flange. |
| WELD           | 1 for closed sections, 2 for open sections | Weld type.  
1 = welding is on one side only except for wide-flange or tee sections, where the web is always assumed to be welded on both sides.  
2 = welding on both sides. For closed sections like a pipe or tube, the welding will be on one side only. |
| WIDTH          | 0.25 times the member length | Effective width of the concrete slab. |
### Design Codes

#### Notes

1. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. See [TR.44 Printing Section Displacements for Members](on page 2846) for details on local displacement.

   If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let \((DX_1, DY_1, DZ_1)\) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, \((DX_2, DY_2, DZ_2)\) represent the deflection values at DJ2 or the end node of the member.

   Compute \[ \Delta = \sqrt{(DX_2 - DX_1)^2 + (DY_2 - DY_1)^2 + (DZ_2 - DZ_1)^2} \]

   Compute \(Length = \) distance between DJ1 and DJ2 or, between start node and end node, as the case may be.

   Then, if \(CAN = 1\), \(dff = L/\Delta\)

   Ratio due to deflection = \(DFF/dff\)

2. If \(CAN = 0\), deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the "Deflection Length" will be equal to the length of the member. However, in some situations, the "Deflection Length" may be different.

   For example, refer to the figure below where a beam has been modeled using four joints and three members. The "Deflection Length" for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Also the straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured. Thus, for all three members here, DJ1 should be "1" and DJ2 should be "4".

![Diagram of beam model](image)

D is equal to the maximum local deflection for members 1, 2, and 3.

PARAMETERS

| DFF 300. ALL
| DJ1 1 ALL
| DJ2 4 ALL

3. If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

4. It is important to note that unless a DFF value is specified, STAAD will not perform a deflection check. This is in accordance with the fact that there is no default value for DFF.

5. A critical difference exists between the parameters UNT/UNB and the parameters LY and LZ. Parameters UNT and UNB represent the laterally unsupported length of the compression flange. It is defined in Chapter F, page...
5-47 of the specifications in the AISC 1989 ASD manual as the distance between cross sections braced against twist or lateral displacement of the compression flange. These parameters are used to calculate the allowable compressive stress ($FC_Z$ and $FC_Y$) for behavior as a beam. Parameters $LY$ and $LZ$ are the unbraced lengths for behavior as a column and are used to calculate the $KL/r$ ratios and the allowable axial compressive stress $FA$.

6. Parameters $SS_Y$ and $CM_Y$ are based upon two values defined in page 5-55, Chapter H of the AISC 9th ed. manual. $SS_Y$ is a variable which allows the user to define whether or not the member is subject to sidesway in the local Y direction. $CM_Y$ is a variable used for defining the expression called $Cm$ in the AISC manual. When $SS_Y$ is set to 0 (which is the default value), it means that the member is subject to sidesway in the local Y direction. When $SS_Y$ is set to 1.0, it means that the member is not subject to sidesway in the local Y direction. The only effect that $SS_Y$ has is that it causes the program to calculate the appropriate value of $CM_Y$. If $SS_Y$ is set to 0 and $CM_Y$ is not provided, STAAD will assume $CM_Y$ as 0.85. If $SS_Y$ is set to 1 and $CM_Y$ is not provided, STAAD will calculate $CM_Y$ from the equation on page 5-55. However, if the user provides $CM_Y$, the program will use that value and not calculate $CM_Y$ at all, regardless of what the user defines $SS_Y$ to be.

![Figure 142: Terms used in calculating slenderness ratios $KL/r$ for local Y and Z axes](image)

7. For a T shape which is cut from a parent I, W, S, M or H shapes, the $PROFILE$ parameter should be assigned a value corresponding to the parent shape. For example, if the T desired is an American WT6, specify W12 for the $PROFILE$ parameter.

D1.B.1.3 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per AISC-89. Code checking is done using the forces and moments at specified sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed the code or have failed; the critical condition of the AISC code (like any of the AISC specifications or compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for a value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member) of forces in the member where the critical condition occurs.
Code checking can be done with any type of steel section listed in D1.A.4 Built-in Steel Section Library (on page 1368) of this manual.

Related Links
- TR.49 Code Checking Specification (on page 2852)
- TR.49.1 Member Selection Specification (on page 2853)
- TR.49.2 Member Selection by Optimization (on page 2854)

D1.B.1.4 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section (i.e., the lightest section) which fulfills the code requirements for the specified member.

The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc. Several parameters are available to guide this selection. If the PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters DMAX and DMIN which limit the maximum and minimum depth of the members. If the PROFILE parameter is provided for specified members, DMAX or DMIN parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections listed in D1.A.4 Built-in Steel Section Library (on page 1368) of this manual. Note that for beams with cover plates, the sizes of the cover plate are kept constant while the beam section is iterated.

Selection of members, whose properties are originally input from a user created table, will be limited to sections in that table.

Member selection cannot be performed on members whose section properties are input as prismatic.

D1.B.1.4.1 Member Selection by Optimization

Steel table properties of an entire structure can be optimized by STAAD.Pro. This optimization method involves the following steps.

1. CHECK CODE ALL
2. Modify the ratios
3. SELECT ALL
4. PERFORM ANALYSIS
5. SELECT ALL

An additional step of grouping may be performed if the FIXED GROUP and GROUP commands are provided (See TR.50 Group Specification (on page 2855)). After the last step, a re-analysis is not automatically performed, so users must ensure that they specify the analysis command following the SELECT OPTIMIZE command.

D1.B.1.4.2 Deflection Check With Steel Design

This facility allows the user to consider deflection as a criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using the three parameters DJ1, DJ2, and DFF which are described in D1.B.1.2 Design Parameters (on page 1409). Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

Related Links
- TR.49.1 Member Selection Specification (on page 2853)
D1.B.1.5 Truss Members

As described in G.8.1 Truss and Tension-or Compression-Only Members (on page 2330), a truss member is capable of carrying only axial forces. So during the design phase, no calculation time (or, matrix bandwidth) is wasted determining the allowable bending or shear stresses. Therefore, if there is any truss member in an analysis (such as a bracing or a strut, etc.), it is advisable to declare it as a truss member rather than as a regular frame member with both ends pinned.

D1.B.1.6 Unsymmetric Sections

For unsymmetric sections like single angles, STAAD.Pro considers the smaller section modulus for calculating bending stresses.

For single angles, the “specification for allowable stress design of single-angle members,” explained in pages 5-309 to 5-314 of the AISC-ASD 9th edition manual has been incorporated.

D1.B.1.7 Composite Beam Design as per AISC-ASD

In G.6.7 Composite Beams and Composite Decks (on page 2328) of this manual, two methods of specifying the properties of a beam as a composite section (I-shaped beam with concrete slab on top) are described. Those members can be designed as composite beams in accordance with the AISC ASD code provisions. If the properties are assigned using the explicit method as defined in Section 1.7.7, the design parameters must be separately assigned. The CMP parameter in particular must be set a value of 1 or 2. If the properties are derived from the composite decks, the design parameters are automatically generated during the deck creation phase, and hence no separate parameters need to be assigned.

Other parameters used in the design of composite members are: DIA, DR1, DR2, FPC, HGT, PLTHICK, PLWIDTH, RBHEIGHT, RBWWIDTH, SHR, THK, and WIDTH. Refer to D1.B.1.2 Design Parameters (on page 1409) for details on design parameters.

```
Example
UNIT INCH
PARAMETER
CODE AISC
BEAM 1 ALL
TRACK 2 ALL
DR1 0.3135 ALL
WID 69.525 ALL
FPC 3.0 ALL
THK 4.0 ALL
CMP 1 ALL
CHECK CODE ALL
SELECT ALL
```

D1.B.1.8 Plate Girders

The requirements of Chapter G – pages 5-51 through 5-53 of the AISC ASD 9th edition manual – are not implemented. Therefore, if the web slenderness ratio \( h/t_w \) of a section exceeds \( \frac{970}{F_y} \), STAAD will not perform a design for that member.
**D1.B.1.9 Tabulated Results of Steel Design**

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

a. **MEMBER** refers to the member number for which the design is performed.

b. **TABLE** refers to the AISC steel section name which has been checked against the steel code or has been selected.

c. **RESULT** prints whether the member has PASsed or FAILed. If the RESULT is FAIL, there will be an asterisk (*) mark in front of the member number.

d. **CRITICAL COND** refers to the section of the AISC code which governed the design.

e. **RATIO** prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.

f. **LOADING** provides the load case number which governed the design.

g. **FX**, **MY**, and **MZ** provide the axial force, moment in local y-axis and moment in local z-axis respectively. Although STAAD does consider all the member forces and moments to perform design, only FX MY and MZ are printed since they are the ones which are of interest, in most cases.

h. **LOCATION** specifies the actual distance from the start of the member to the section where design forces govern.

i. If the parameter **TRACK** is set to 1.0, the program will block out part of the table and will print the allowable bending stresses in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV), all in kips per square inch. In addition, member length, area, section moduli, governing KL/r ratio and CB are also printed.

j. In the output for **TRACK 2.0**, the items Fey and Fez are as follows:

\[
F_{ey} = 12n^2E/[23 \cdot (K_yL_y/r_y)^2] \\
F_{ez} = 12n^2E/[23 \cdot (K_zL_z/r_z)^2]
\]

---

**Example Output**

<table>
<thead>
<tr>
<th>Y</th>
<th>PROPERTIES IN INCH UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>---</td>
<td>------------------------</td>
</tr>
<tr>
<td>MEMBER 5</td>
<td>AISC SECTIONS ST W14X30</td>
</tr>
<tr>
<td>DESIGN CODE</td>
<td>AISC-1989</td>
</tr>
<tr>
<td>&lt;---LENGTH (FT)= 10.00 ---&gt;</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARAMETER IN KIP INCH</th>
<th>L1 L1 STRESSES IN KIP INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y= 80.64</td>
<td>L1 FA = 15.11</td>
</tr>
<tr>
<td>KL/R-Z= 20.93</td>
<td>L1 fa = 0.30</td>
</tr>
<tr>
<td>UNL = 120.00</td>
<td>L1 FCZ = 19.95</td>
</tr>
<tr>
<td>CB = 1.00</td>
<td>L1 FTZ = 21.60</td>
</tr>
<tr>
<td>CMY = 0.85</td>
<td>L1 FCY = 27.00</td>
</tr>
</tbody>
</table>
## Design

### D. Design Codes

<table>
<thead>
<tr>
<th>CMZ</th>
<th>FYLD</th>
<th>NSF</th>
<th>DFF</th>
<th>dff=</th>
<th>(KL/R)max =</th>
<th>FTY =</th>
<th>fbz =</th>
<th>fby =</th>
<th>Fey =</th>
<th>Fez =</th>
<th>FV =</th>
<th>fv =</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.85 + L1</td>
<td>36.00</td>
<td>0.85 +</td>
<td>0.00 -9.1</td>
<td>0.00</td>
<td>80.64</td>
<td>27.00</td>
<td>46.60</td>
<td>0.00</td>
<td>22.97</td>
<td>340.99</td>
<td>14.40</td>
<td>4.78</td>
</tr>
</tbody>
</table>

### MAX FORCExx MOMENT SUMMARY (KIP-FEET)

<table>
<thead>
<tr>
<th>VALUE</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOMENT-Y</th>
<th>MOMENT-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-2.7</td>
<td>16.5</td>
<td>0.0</td>
<td>0.0</td>
<td>163.3</td>
</tr>
<tr>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>10.0</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

### DESIGN SUMMARY (KIP-FEET)

<table>
<thead>
<tr>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FAIL</td>
<td>AISC- H2-1</td>
<td>2.335</td>
<td>1</td>
</tr>
<tr>
<td>2.66 T</td>
<td>0.00</td>
<td>-163.32</td>
<td>10.00</td>
</tr>
</tbody>
</table>

---

### D1.B.2 Castellated Beams

STAAD.Pro comes with the non-composite castellated beam tables.

![Castellated beam elevation](image)

**Figure 143: Castellated beam elevation**

According to the manufacturer, castellated beams are manufactured by cutting a wide flange along the web in a "zig-zag" pattern, offsetting the two halves, and welding the two halves together, as shown in the next figure. As a result, the underlying steel section is a wide flange (W shapes) in the AISC table or a B shape. STAAD currently supports only the ones derived from W shapes.
Figure 144: Manufacturing process for castellated beams: A) Cut hexagonal line, B) Stagger top and bottom sections, and C) discard waste at ends

Related Links

- D1.A.4.3 Castellated Beams Section Sizes (on page 1371)

D1.B.2.1 Analysis and Design Criteria

The local axis system (local X, local Y and local Z) of a castellated beam is identical to that for a wide flange, and is shown in G.4.2 Local Coordinate System (on page 2297).

It is important recognize that there are two basic issues to be understood with regard to these members a) analysis b) steel design

First, the design issues because only then will their relationship with the analysis issues become apparent. Design of a castellated beam is done only for FY (shear along the web) and MZ (moment about the major axis which is the Z axis). If at the start of the design process, the program detects that the beam has axial force (FX),
shear along local-Z (FZ), torsion (MX) or moment about the minor axis (MY), design of that member will be terminated.

Next is how these design limitations have a bearing on the analysis issues. If you intend to design these members, as a result of the above restrictions, he/she must model it in such a way that none of the 4 unacceptable degrees of freedom end up with a non-zero value anywhere along the length of the member. That means, if the member ends are defined as supports, the support conditions must be defined with the above in mind. Similarly, if the castellated member is attached to other members, its end conditions (MEMBER RELEASES) must be modeled taking the above facts into consideration.

The design limitations also have a bearing on the type of loads that are applied to the member. Loads which cause any of the above-mentioned four degrees of freedom to end up with a non-zero value will cause the member design to be terminated.

However, if you wish to only analyze the structure, and are not interested in performing a steel design, the above described restrictions for supports, member end conditions or loading are not applicable.

The design method is the allowable stress method, using mainly the rules stated in the AISC ASD 9th edition code. Only code checking is currently available for castellated beams. Member selection is not.

**Note:** STAAD.Pro does not multiply the analysis moment by 1.7 for ASD method. It is up to you to multiply the dead and live loads by 1.7 in load combination and using this load case in design. The reason is that if program internally multiplies the analysis moment by 1.7 for ASD method (it is 1.2 for dead and 1.6 for live loads for LRFD method) then you must ensure that the analysis moment is the unfactored moment. If by mistake the 1.7 factor is used during load combination and the I-beam with web opening is designed with this load, the program will further increase the load by 1.7. Hence, it has been intentionally left to your engineering judgment to use whatever load factor you see fit before designing I-beam with opening with that factored load case.

**D1.B.2.2 Design Parameters**

The following table contains a list of parameters and their default values.

**Table 92: American Castellated Beam Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Cb value used for computing allowable bending stress per Chapter F of the AISC specifications.</td>
</tr>
<tr>
<td>CMZ</td>
<td>0.85</td>
<td>Cm value in local z-axis. Used in the interaction equations in Chapter H of the AISC specifications.</td>
</tr>
</tbody>
</table>
### Design

#### D. Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EOPEN</td>
<td>1.5e + b</td>
<td>Distance from the center of the last hole to the end of the member. 1.5e + b is the minimum allowable value. Any value greater than or equal to this minimum will be used by the program. See Figure 2.13 for the definition of e and b.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 ksi</td>
<td>Yield stress of steel.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible maximum ratio of actual load to section capacity. Any input value will be used to change the right hand side of the governing interaction equations in Chapter H and elsewhere in the AISC specifications.</td>
</tr>
<tr>
<td>SOPEN</td>
<td>1.5e + b</td>
<td>Distance from the start of the member to the center of the first hole. 1.5e + b is the minimum allowable value. Any value greater than or equal to this minimum will be used by the program. See Figure 2.13 for the definition of e and b.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>Used to control the level of description of design output. 0 = Detailed output suppressed 1 = Detail output included</td>
</tr>
<tr>
<td>UNL</td>
<td>Member Length</td>
<td>Unsupported length of compression flange for calculating allowable bending stress.</td>
</tr>
</tbody>
</table>

**D1.B.2.3 Design Procedure**

**Cross-Section Checks**

The first check that is carried out is a verification whether the member properties satisfy certain basic requirements. If the member fails these checks, the remainder of the checks are not performed.

The cross section checks are the following:
1. Web Post Width (e) should be at least 3.0 inches
2. Tee Depth (d_{T\text{-top}} and d_{T\text{-bot}}) should be greater than the thickness of flange plus one inch.
3. Angle θ should be between 45 and 70 degrees.
4. In order for the program to determine the number of holes which are admissible for the beam, the parameters SOPEN and EOPEN need to be assigned. In the figure above, there is a term shown as S. This value is part of the section tables supplied with STAAD.Pro, so it retrieves that value from there. It then computes the number of holes, and the remainder of the terms shown in the above diagram.
5. SOPEN and EOPEN (see the parameter table shown earlier) have to be at least 1.5e + b, with “e” and “b” as shown in the earlier figure. If you inputs a value less than these minima, the minimum values are used.

References

Design of castellated beams in STAAD.Pro is based on the information gathered from the following sources:

2. AISC 9th edition manual – Allowable stress design

Checking the member for adequacy in carrying the applied loading

This consists of five different checks:

1. Global Bending
2. Vierendeel Bending
3. Horizontal Shear
4. Vertical Shear
5. Web Post Buckling
### Table 93: Cross section considered for limit states

<table>
<thead>
<tr>
<th>Design For</th>
<th>Section Considered in the Design (shown with the vertical dotted lines)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vierendeel Bending</td>
<td></td>
</tr>
<tr>
<td>Global Bending</td>
<td></td>
</tr>
<tr>
<td>Vertical Shear</td>
<td></td>
</tr>
<tr>
<td>Horizontal Shear</td>
<td></td>
</tr>
<tr>
<td>Web Post Buckling</td>
<td></td>
</tr>
</tbody>
</table>

1. **Global Bending:**
   Global bending check is done at the web post section. This is the region of the member where the full cross section is active, without interference of the holes.

   The actual bending stress is computed at the middle of the web post location and is obtained by dividing the moment by the section modulus of the full section.

   For computing the allowable bending stress, the compactness of the section is first determined in accordance with Table B5.1 in the Chapter B of the AISC 9th edition specifications. The rules applicable to I-shaped sections are used for this. Following this, the allowable bending stress is computed per chapter F of the same.

   The ratio is computed by dividing the actual stress by the allowable stress.

2. **Vierendeel Bending:**
   This is checked at the middle of the hole locations. The effective cross section at these locations is a Tee. The overall moment (Mz) at the span point corresponding to the middle of the hole is converted to an axial force and a moment on the Tee.

   The actual stress is computed at the top and bottom of each Tee section.

   \[ f_a = \frac{M}{(d_{\text{effect}} \times A_t)} \]

   where

   \[ A_t = \text{area of the Tee section} \]

   \[ f_b = \frac{V \times e \times a}{(2\times S)} \]

   where
\[ a = \text{the area factor. For the top Tee section, } a = \frac{\text{Area of Top Tee}}{(\text{Area of Top Tee} + \text{Area of Bottom Tee})} \]

**Allowable Stresses for vierendeel bending:**

- **Axial Stress:** The allowable axial stress is computed as per the Chapter E of the AISC specifications. The unsupported length for column buckling is equal to \( e \).
- **Bending Stress:** The allowable bending stress is computed for the top and bottom Tee section as per the Chapter F of the AISC manual.

The axial stress plus bending stress is computed at the top and bottom of each tee section. If it is compressive then it is checked against equations H1-1 and H1-2 of Chapter H of the AISC manual. If it is tensile then it is checked against equation H2-1.

3. **Horizontal Shear**
   
   Allowable Shear stress is computed as \( 0.4 F_y \).
   
   Actual Stress: Please refer to pages 4.7-8 and 4.7-9 of reference #1.

4. **Vertical Shear**
   
   Allowable Shear stress is computed as \( 0.4 F_y \).
   
   The actual shear stress is computed at the middle of the web post location.

5. **Web Post buckling**
   
   Refer to pages 1202-1207 of reference #3.

**D1.B.2.4 General Format**

The command syntax in the STAAD input file for assigning castellated beams is:

```
MEMBER PROPERTY AMERICAN
```

**Member-list**

```
Table ST section-name
```

**Example**

```
MEMBER PROPERTY AMERICAN
2 TABLE ST CB12X28
```

Assigning Design parameters

Under the **PARAMETERS** block on input, the code name must be specified as:

```
CODE AISC CASTELLATED
```

**Example**

```
PARAMETER
CODE AISC CASTELLATED
UNL 0.01 MEMB 25 31
FYLD 50 MEMB 25 31
SOPEN 11.124 MEMB 25 31
...
CHECK CODE MEMB 25 31
```
## D1.B.2.5 Steel Design Output

The following is a typical TRACK 2 level output page from a STAAD output file.

<table>
<thead>
<tr>
<th>STAAD.PRO CODE CHECKING - (AISC CASTELLATED)</th>
<th>v1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALL UNITS ARE - Kip and Inches (UNLESS OTHERWISE NOTED)</td>
<td></td>
</tr>
</tbody>
</table>

### Castellated Steel Design for Member 2

Section Name ST CB27X40

---

#### Design Results

Design Status: Pass Critical Ratio: 0.96

- **Check for Global Bending**
  - Load = 3
  - Section = 260.874
  - $F_y = 0.76$, $M_z = -3020.39$
  - $F_{b\ top} = 33.00$, $F_{b\ Bot} = 33.00$
  - $fb = 26.83$
  - Ratio = 0.81

- **Check for Vierendeel Bending**
  - Load = 3
  - Section = 214.624
  - $F_y = 4.61$, $M_z = -2894.76$
  - $F_a = 29.91$, $F_b = 30.00$
  - $K_{lr} = 1.46$, $F_e = 69606.88$
  - $fa = 26.08$, $fb = 2.79$
  - Ratio = 0.96

- **Check for Vertical Shear**
  - Load = 3
  - Section = 0.000
  - $F_y = 22.50$, $M_z = 0.00$
  - $F_v = 20.00$, $f_v = 2.62$
  - Ratio = 0.13

- **Check For Horizontal Shear ( Web Post )**
  - Load = 3
  - Section = 519.874
  - $F_y = -20.82$, $M_z = -415.10$
  - $F_v = 20.00$, $f_v = 14.73$
  - Ratio = 0.74

- **Check for Web Post Buckling**
  - Load = 3
  - Section = 519.874
  - $F_y = -20.82$, $M_z = -415.10$
  - Mallow = 141.32, Mact = 189.47
  - Ratio = 0.75

Viewing the design results in the GUI
1. After the analysis and design is completed, double click on the castellated member in the STAAD.Pro View window. The Beam dialog box.

2. Select the Castellated Beam Design tab.

![Beam dialog box Castellated Beam Design tab](image)

*Figure 146: Beam dialog box Castellated Beam Design tab*

3. (Optional) Click Print to create a hard copy of the Castellated Beam data for this beam.

4. Click Close. The dialog closes.

**D1.B.2.6 Example**

The following is an example STAAD Input for a portal frame with a castellated beam.
**Tip:** You can copy the input code below and paste into the STAAD Editor or into a plain text editor program and save as an .STD file for use in STAAD.Pro.

```
STAAD PLANE EXAMPLE PROBLEM FOR
*CASTELLATED BEAM DESIGN
UNIT FT KIP
JOINT COORDINATES
1 0. 0. ; 2 45 0
3 0 15 ; 4 45 15
MEMBER INCIDENCE
1 1 3; 2 3 4; 3 4 2
MEMBER PROPERTY AMERICAN
2 TA ST CB27x40
1 3 TA ST W21X50
UNIT INCH
CONSTANTS
E STEEL ALL
DEN STEEL ALL
POISSON STEEL ALL
MEMBER RELEASE
2 START MX MY MZ
2 END MY MZ
UNIT FT
SUPPORT
1 2 FIXED
LOADING 1 DEAD AND LIVE LOAD
MEMB LOAD
2 UNI Y -0.4
LOADING 2 WIND FROM LEFT
MEMBER LOAD
2 UNI Y -0.6
LOAD COMB 3
1 1.0 2 1.0
```
D1.B.3 Design of Beams with Web Openings

Design of steel members with web openings per AISC Steel Design Guide 2 - ASD specifications may be performed in STAAD for members whose yield strength is 65 ksi or less.

Note: The web openings are given consideration only during the design phase. The reduction in section properties caused by the presence of the openings is not considered automatically during the analysis phase. Hence, the analysis is performed as if the full section properties are effective for such members.

During the design process, the program first determines the utilization ratio (U.R.) at the location of the opening as though it is an unreinforced opening. If the U.R. is less than 1.0, the member is presumed to have passed the requirements at that location. If the U.R. exceeds 1.0, then it determines the U.R. as though it is a reinforced opening. If it fails this too, the cause of the failure along with the associated numerical values is reported.

D1.B.3.1 Description

The following are the salient points of the design process.

A. Only a code check operation is permitted on members with web openings. The MEMBER SELECTION process will not be performed if a web opening is specified for the member.

B. The CODE CHECK operation is performed at the following locations along the member span:

   a. The 13 equally spaced points along the member span customary with the BEAM 1 parameter

      or

      The section locations specified using the SECTION command, if the BEAM parameter is set to 0.0

      or

      The 2 member ends if BEAM parameter is set to 0, and the SECTION command is not specified.

   b. At the web openings locations defined using the RHOLE and CHOLE parameters (Refer to Table 2.18-1 below).

If any of the locations defined under (a) above happen to coincide with those in (b), such locations are designed as places where openings are located, and not as an unperforated section location.

The utilization ratio (U.R.) is determined for all the locations in (a) above, as well as all the locations in (b) above. The highest value among these locations is deemed critical from the design standpoint.
The design output consists of the critical value obtained from checking the locations under (a), and each of the locations under (b).

The critical location among those in (b) is not displayed in the post-processing pages of the program such as the Beam-Unit Check page, or the Member Query-Steel Design tab.

Members declared as TRUSS (trusses) or TENSION (tension-only) are not designed for web openings.

**D1.B.3.2 Design steps for Steel Beam with Web Opening**

At the location of web holes, the capacity of the section is determined using the rules explained below.

![Figure 148: Opening configurations for steel beams with unreinforced opening](image)

![Figure 149: Opening configurations for steel beams with reinforced opening](image)

**Section Properties Required:**

- \( A_s \) = Cross-sectional area of steel in the unperforated member (at a section along the beam where there is no opening)
- \( d \) = Depth of beam
- \( t_w \) = Thickness of web
- \( b_f \) = Width of flange
- \( t_f \) = Thickness of flange
- \( Z \) = Plastic section modulus of member without opening
- \( J \) = Torsional constant of the beam
- \( L \) = Length of the member
- \( L_b \) = Unbraced length of compression flange
Opening Information Required:

\[ e = \text{ Eccentricity of opening, specified using the HECC parameter (distanced from mid-depth of beam to mid-depth of the opening)} = | e | \]

(sign convention for eccentricity: upward eccentricity + e, downward eccentricity - e)

Loading:

\[ V_u = \text{ Factored shear at different opening locations} \]
\[ M_u = \text{ Factored bending moment at different opening locations} \]

Calculated Parameters

Circular Opening Properties:

Unreinforced Opening Properties:

\[ h_o = D_o \text{ for bending} \]
\[ h_o = 0.9 D_o \text{ for shear} \]
\[ a_o = 0.45 D_o \]

Reinforced Opening Properties:

\[ h_o = D_o \text{ for bending and shear} \]
\[ a_o = 0.45 D_o \]

Tee Properties:

Refer to Figure 2.17(a), above

\[ s_t = \text{ Depth of top tee} = d/2 - (h_o / 2 + e) \]
\[ s_b = \text{ Depth of bottom tee} = d/2 - (h_o / 2 - e) \]

Reinforcement Properties:

\[ A_r = \text{ Area of reinforcement on each side of the opening} \]
\[ t_r = \text{ Thickness of reinforcement bar} \]
\[ b_r = \text{ Width of reinforcement bar} \]
\[ D_r = \text{ Depth of reinforcement bar} \]

**D1.B.3.3 Calculation Steps**

1. Check for local buckling of compression flange and reinforcement (if any)

   (AISC Design Guide 2 for Web Openings: section 3.7.a.1)

   \[ F_1 = b_f / 2t_f \]

   \[ F_2 = b_r / t_r \]

   Limiting width to thickness ratio, \( B_1 = 65 / \sqrt{F_y} \)

   \( F_1 \) and \( F_2 \) must not exceed \( B_1 \).

2. Check for web buckling

   (AISC Design Guide 2 for Web Openings: section 3.7.a.2)

   \[ W_1 = (d - 2t_f) / t_w \]
**W**

3. Check for opening dimensions to prevent web buckling

(AISC Design Guide 2 for Web Openings: section 3.7.a.2 & section 3.7.b.1)

a. Limit on \(a_o / h_o\) as given below,

If \(W_1 \leq 420/\sqrt{F_y}\) then web qualifies as stocky,

\[ a_o / h_o \text{ must not exceed } 3.0, \]

If \(W_1 > 420/\sqrt{F_y}\) but \(\leq 520/\sqrt{F_y}\) then

\[ a_o / h_o \text{ must not exceed } 2.2, \]

b. \(h_o / d\) must not exceed 0.7

c. The opening parameter, \(p_o = (a_o / h_o) + (6 h_o / d)\) must not exceed 5.6

4. Check for tee dimensions

(AISC Design Guide 2 for Web Openings: section 3.7.b.1)

a. The maximum depth of the web opening is governed by the following rules:

Depth \(s_t \geq 0.15d, s_b \geq 0.15d\)

b. Aspect ratios of the tees \((v = a_0 / s)\) should not be greater than 12

\[ a_0 / s_t \leq 12, a_0 / s_b \leq 12, \]

5. Check for buckling of tee-shaped compression zone

(AISC Design Guide 2 for Web Openings: section 3.7.a.3)

The tee, which is in compression, is investigated as an axially loaded column. For unreinforced members this is not done when the aspect ratio of the tee is less than or equal to four:

\[ F_y t_w d/\sqrt{3} \leq 4 \]

For reinforced openings, this check is only done for large openings in regions of high moment.

6. Calculation for Maximum Moment Capacity, \(M_m\)

(AISC Design Guide 2 for Web Openings: section 3.5.a)

For unperforated section

\[ M_p = F_y Z \]

\[ \Delta A_s = h_o t_w - 2A_r \]

Unreinforced Opening:

\[ M_m = M_p \left[ 1 - \frac{\Delta A_s \left( h_o / 4 + e \right)}{Z} \right] \]

Reinforced Opening:

a. If \(t_w e < A_r\)

\[ M_m = M_p \left[ 1 - \frac{t_w \left( h_o^2 / 4 + h_o e - e^2 \right) - A_r h_o}{Z} \right] \leq M_p \]

b. If \(t_w e \geq A_r\)
Design
D. Design Codes

\[
M_m = M_p \left[ 1 - \frac{\Delta A_p \left( \frac{h_o}{4} + e - \frac{A_p}{2t_w} \right)}{Z} \right] \leq M_p
\]

7. Calculation for Maximum Shear Capacity, \( V_m \)

(AISC Design Guide 2 for Web Openings: section 3.6.a)

\( V_{pb} \) and \( V_{pt} \) = Plastic shear capacity of the web of the tee

\[
V_{pb} = F_y t_w s_b \sqrt{3}
\]

\[
V_{pt} = F_y t_w s_t \sqrt{3}
\]

The values of aspect ratios \( v_b \) and \( v_t \) and factors \( \mu_b \) and \( \mu_t \) (which appear in the equations shown below) are different for reinforced and unreinforced openings.

Unreinforced Opening:

For the bottom tee, \( v_b = a_o / s_b \) and \( \mu_b = 0 \)

For the top tee, \( v_t = a_o / s_t \) and \( \mu_t = 0 \)

Reinforced Opening:

\( s_{t1} \) and \( s_{b1} \) are used to calculate for \( n \) reinforced opening.

\[
s_{t1} = s_t \cdot A_t / (2b_f)
\]

\[
s_{b1} = s_b \cdot A_t / (2b_f)
\]

\( P_r \) = Force in reinforcement along edge of opening = \( F_y A_t \leq F_y t_w a_o / (2\sqrt{3}) \)

\( d_{rt} \) and \( d_{rb} \) = Distance from outside edge of flange to centroid of reinforcement.

\[
d_{rt} = s_t - \frac{1}{2} t_r
\]

\[
d_{rb} = s_b - \frac{1}{2} t_r
\]

For the bottom tee, \( v_b = a_o / s_{b1} \) and \( \mu_b = 2 P_r d_{rb} / (V_{pb} s_b) \)

For the top tee, \( v_t = a_o / s_{t1} \) and \( \mu_t = 2 P_r d_{rt} / (V_{pt} s_t) \)

General Equations:

Using equations given below for \( \alpha_{vb} \) and \( \alpha_{vt} \)

\( \alpha_{vb} = \) Ratio of nominal shear capacity of bottom tee, \( V_{mb} \) to plastic shear capacity of the web of the tee = \( (\sqrt{6} + \mu_b) / (v_b + \sqrt{3}) \leq 1 \)

\( \alpha_{vt} = \) Ratio of nominal shear capacity of top tee, \( V_{mt} \) to plastic shear capacity of the web of the tee = \( (\sqrt{6} + \mu_t) / (v_t + \sqrt{3}) \leq 1 \)

\[
V_{mb} = V_{pb} \alpha_{vb}
\]

\[
V_{mt} = V_{pt} \alpha_{vt}
\]

\( V_m \) = Maximum nominal shear capacity at a web opening = \( V_{mb} + V_{mt} \)

8. Check against Maximum Shear Capacity \( V_m \)

(AISC Design Guide 2 for Web Openings: section 3.7.a.2)

\( V_p \) = Plastic shear capacity of unperforated web = \( F_y t_w d / \sqrt{3} \)

If \( W_1 \leq 420/\sqrt{F_y} \), \( V_m \) must not exceed \( 2/3 V_p \)
If \( W_1 > \frac{420}{\sqrt{F_y}} \) but \( \leq \frac{520}{\sqrt{F_y}} \), \( V_m \) must not exceed \( 0.45 V_p \)

9. Check against Moment Shear Interaction

AISC Design Guide 2 for Web Openings: section 3.2 & section 3.4)

\[
R1 = \frac{V_u}{V_m} \leq 1.0 \\
R2 = \frac{M_u}{M_m} \leq 1.0 \\
R1^3 + R2^3 \leq R^3, R \leq 1.0
\]

10. Corner Radii (for reinforced opening only)

(AISC Design Guide 2 for Web Openings: section 3.7.b.2)

Minimum radii = the greater of \( 2t_w \) or \( 5/8 \) inch

11. Calculation of length of fillet weld (for reinforced opening only)

(AISC Design Guide 2 for Web Openings: section 3.7.b.5)

For reinforcing bars on both sides / on one side of the web:

Fillet welds should be used on both sides of the reinforcement on extensions past the opening. The required strength of the weld within the length of the opening is,

\[
R_{wr} = 2P_r
\]

Where:

\( R_{wr} = \text{Required strength of the weld} \)

The reinforcement should be extended beyond the opening by a distance

\[
L_1 = \frac{a_o}{4} \text{ or } L_1 = \frac{A_r\sqrt{3}}{(2t_w)}
\]

whichever is greater, on each side of the opening. Within each extension, the required strength of the weld is

\[
R_{wr} = F_y A_r
\]

Additional requirements for reinforcing bars on one side of the web:

\( A_f = \text{area of flange} = b_f t_f \)

a. \( A_r \leq \frac{A_f}{3} \)

b. \( a_o / h_o \leq 2.5 \)

c. \( V1 = s_t / t_w \text{ or } V2 = s_b / t_w \)

\( V1 \) and \( V2 \leq \frac{140}{\sqrt{F_y}} \)

d. \( \frac{M_u}{(V_u d)} \leq 20 \)

12. Calculation for spacing of openings

(AISC Design Guide 2 for Web Openings: section 3.7.b.6)

Rectangular Opening:

\[
S \geq h_o \\
S \geq a_o \left( \frac{V_u}{V_p} \right) / \left[ 1 - \left( \frac{V_u}{V_p} \right) \right]
\]

Circular Opening:

\[
S \geq 1.5 D_o \\
S \geq D_o \left( \frac{V_u}{V_p} \right) / \left[ 1 - \left( \frac{V_u}{V_p} \right) \right]
\]
13. Check for deflection

The deflection check is performed using the approximate procedure described in section 6.2 of the AISC Design Guide 2 for Web Openings.

**D1.B.3.4 General Format**

<table>
<thead>
<tr>
<th>RHOLE</th>
<th>r1 r2 r3 Memb &lt;list&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHOLE</td>
<td>c1 c2 c3 Memb &lt;list&gt;</td>
</tr>
</tbody>
</table>

Where:

\[ r_1, r_2, \text{ and } r_3 \text{ and } c_1, c_2, \text{ and } c_3 \text{ are the section locations of three rectangular and three circular openings respectively, along the length of the member in ascending order from the start of the member (i.e., } r_1 < r_2 < r_3 \text{ and } c_1 < c_2 < c_3) \]

**Notes**

The maximum number of openings allowed for each member is three. Thus there can be three rectangular openings, three circular openings, or a combination of three rectangular and circular openings for each member.

<table>
<thead>
<tr>
<th>RDIM</th>
<th>[l1 d1] [l2 d2] [l3 d3] Memb &lt;list&gt;</th>
</tr>
</thead>
</table>

Where \( l_1, l_2, \text{ and } l_3 \text{ are the three different lengths and } d_1, d_2 \text{ and } d_3 \text{ are the three different depths of the rectangular openings.} \)

<table>
<thead>
<tr>
<th>CDIA</th>
<th>d1 d2 d3 Memb &lt;list&gt;</th>
</tr>
</thead>
</table>

Where \( d_1, d_2, \text{ and } d_3 \text{ are the three different diameters of the circular openings.} \)

<table>
<thead>
<tr>
<th>HECC</th>
<th>e1 e2 e3 Memb &lt;list&gt;</th>
</tr>
</thead>
</table>

If the eccentricity of the opening is in the negative local Y-axis of the member the sign should be negative.

<table>
<thead>
<tr>
<th>ELECTRODE</th>
<th>f Memb &lt;list&gt;</th>
</tr>
</thead>
</table>

Where \( f \) is the weld material used to calculate size and length of fillet weld required to connect reinforcing bars on beam web at opening.

**Example**

<table>
<thead>
<tr>
<th>UNIT</th>
<th>INCH PARAMETER RHOLE 0.4 0.6 Memb 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>RDIM</td>
<td>10.0 5.0 20.0 10.0 Memb 5 CHOLE 0.8 Memb 5 CDIA 10.0 Memb 5 ELECTRODE 3 Memb 5</td>
</tr>
</tbody>
</table>

The above example shows that member 5 contains two rectangular openings at sections 0.4 and 0.6 whereas one circular opening is located at section 0.8 of the member. The dimensions of rectangular openings are 10.0 × 5.0 and 20.0 × 10.0 inch respectively whereas diameter of circular opening is 10.0 inch.

**1B.3.5 Example 1: Design of Steel Beam with Web Opening**

This example illustrates the design procedure applied at the location of the web opening. The design at locations where openings are not present is not shown in this example.

**Problem**

Design Method : ASD
Beam No : 5
Beam Section : W21X50

d = 20.83 in
\( t_w = 0.38 \) in
\( b_f = 6.53 \) in
\( t_f = 0.535 \) in
\( Z = 110 \) in\(^3\)
\( L = 10 \) ft

Weld Properties : E90XX
\( F_{yw} = 90 \) ksi
Weld Stress = 54 ksi

Opening Type: Rectangular
Number of openings : 1

Section location of Opening : 0.6
\( a_0 = 20 \) in
\( h_0 = 10 \) in
\( e = 0 \) in ( concentric opening )
\( F_y = 36 \) ksi

Capacity check at web hole assuming an unreinforced opening:

Loading :
\( V_u = 34.46 \) kip
\( M_u = 2,510.7 \) kip-inch

Tee Properties :
\( s_t = 5.415 \) in
\( s_b = 5.415 \) in

Calculations

Check for local buckling of compression flange :

width to thickness ratio \( F_1 = \frac{b_f}{t_f} = 6.1028 \)

limiting ratio = \( \frac{65}{\sqrt{F_y}} = 86.667 > F_1 \) Hence O.K

Check for web buckling :

width to thickness ratio of web \( W_1 = \frac{(d - 2t_f)}{t_w} = 52 \)

limiting ratio = \( \frac{520}{\sqrt{F_y}} = 86.667 > W_1 \) Hence O.K
Check for opening dimensions to prevent web buckling:

Since \( W_1 \leq \frac{420}{\sqrt{F_y}} = 70 \), \( \alpha_0 \) / \( h_0 \) = 2 < 3.0

\( h_0 / d = 0.4801 < 0.7 \)

Opening parameter \( p_0 = \frac{a_0}{h_0} + \frac{6h_0}{d} = 4.8805 < 5.6 \) Hence OK

Check for Tee dimension:

\[ s_t = \frac{d}{2} - \left( \frac{h_0}{2} + e \right) = 5.415 \text{ in.} \geq 0.15d = 3.1245 \text{ in.} \]

\[ s_b = \frac{d}{2} - \left( \frac{h_0}{2} - e \right) = 5.415 \text{ in.} \geq 0.15d = 3.1245 \text{ in.} \]

Aspect Ratio:

\[ v_t = \frac{\alpha_0}{s_t} = 3.6934 < 12.0 \]

\[ v_b = \frac{\alpha_0}{s_b} = 3.6934 < 12.0 \] Hence OK

Calculation of Maximum Moment Capacity:

For unperforated section \( M_p = F_yZ = 3,960 \text{ kip-inch} \)

\[ \Delta A_s = h_0 t_w = 3.8 \text{ in}^2 \]

\[ M_m = M_p \left[ 1 - \frac{\Delta A_s (h_0 / 4 + e)}{Z} \right] = 3, 618 \text{ kip-in} \leq M_p \] Hence OK

Calculation of Maximum Shear Capacity

\[ V_{pb} = \frac{F_y t_w s_b}{\sqrt{3}} = 42.768 \text{ kips} \]

\[ V_{pt} = \frac{F_y t_w s_t}{\sqrt{3}} = 42.768 \text{ kips} \]

For unreinforced opening:

\( \mu_b = 0, \mu_t = 0 \)

Ratio of nominal shear capacity of tees:

\[ \alpha_{vb} = \frac{\sqrt{6} + \mu_b}{s_b + \sqrt{3}} = 0.4515 \leq 1.0 \]

\[ \alpha_{vt} = \frac{\sqrt{6} + \mu_t}{s_t + \sqrt{3}} = 0.4515 \leq 1.0 \]

Hence OK
\[ V_{mb} = V_{pb} \alpha_{vb} = 19.309 \text{ kips} \]
\[ V_{mb} = V_{pb} \alpha_{vb} = 19.309 \text{ kips} \]
\[ V_m = V_{mb} + V_{mb} = 38.618 \text{ kips} \text{ Hence OK} \]

Check against Maximum Shear capacity:
\[ V_p = \frac{F_y t w d}{\sqrt{3}} = 164.52 \text{ kips} \]

Since \( W_1 \leq \frac{420}{\sqrt{F_y}} \), \( V_m \leq \frac{2}{3} V_p = 109.68 \text{ kips} \text{ Hence OK} \)

Check against Moment Shear Interaction:
\[ R_1 = \frac{V_u}{\phi V_m} = 0.8923 \leq 1.0 \]
\[ R_2 = \frac{M_u}{\phi M_m} = 0.6939 \leq 1.0 \]
\[ R = \sqrt[3]{R_1^3 + R_2^3} = 1.0147 > 1.0 \]

Not OK... Try with a Reinforced web opening.

Capacity check at web hole assuming a reinforced opening:
Reinforcement should be selected to reduce \( R \) to 1.0

Let us assume,

- Thickness of Reinforcement \( t_r = 0.1875 \text{ in} \)
- Width of Reinforcement \( b_r = 0.25 \text{ in} \)

Check for local buckling of compression flange:

width to thickness ratio of web reinforcement \( F_2 = \frac{b_r}{t_r} = 1.3333 \)

limiting ratio \( = \frac{65}{\sqrt{F_y}} = 10.833 > F_2 \text{ Hence O.K} \)

Area of Reinforcement \( A_r = t_r b_r = 0.0469 \text{ in}^2 \)

Calculation of Maximum Moment Capacity:
For unperforated section \( M_p = F_y Z = 3.960 \text{ kip-inch} \)

\[ \Delta A_s = h_0 t_w - 2 A_r = 3.7063 \text{ in}^2 \]

Since \( t_w e = 0 < A_r \),
\[ M_m = M_p \left[ 1 - \frac{t_w \left( h_0 \sqrt{2} / 4 + h_0 e - e^2 \right) - A_r h_0}{Z} \right] = 3, 634.88 \text{ kip-in} \leq M_p \text{ Hence OK} \]

Calculation of Maximum Shear Capacity:
\[ V_{pb} = \frac{F_y t w b}{s_b} = 42.768 \text{kips} \]
\[ V_{pt} = \frac{F_y t w t}{s_t} = 42.768 \text{kips} \]
\[ s_{t1} = s_t - \frac{A_r}{2b_f} = 5.4114 \text{in} \]
\[ s_{b1} = s_b - \frac{A_r}{2b_f} = 5.4114 \text{in} \]

\[ P_r = F_y A_r = 1.6875 \leq \frac{F_y t w a_0}{2\sqrt{3}} = 78.982 \text{kips} \text{ Hence Ok} \]
\[ d_{rt} = s_t - \frac{1}{2} t_r = 5.3213 \]
\[ d_{rb} = s_b - \frac{1}{2} t_r = 5.3213 \]

\[ v_t = \frac{a_0}{s_{t1}} = 3.6959 < 12.0, \mu_t = \frac{2P_r d_{rt}}{V_{pt} s_t} = 0.0775 \]
\[ v_b = \frac{a_0}{s_{b1}} = 3.6959 < 12.0, \mu_b = \frac{2P_r d_{rb}}{V_{pb} s_b} = 0.0775 \]

\[ \alpha_{vb} = \frac{\sqrt{6} + \mu_b}{v_b + \sqrt{3}} = 0.4656 \leq 1.0 \]
\[ \alpha_{vt} = \frac{\sqrt{6} + \mu_t}{v_t + \sqrt{3}} = 0.4656 \leq 1.0 \]

Hence OK

\[ V_{mb} = V_{pb} \alpha_{vb} = 19.911 \text{kips} \]
\[ V_{mt} = V_{pt} \alpha_{vt} = 19.911 \text{kips} \]
\[ V_m = V_{mb} + V_{mt} = 39.823 \text{kips} \]

Check against Maximum Shear capacity :

\[ V_p = \frac{F_y t w d}{\sqrt{3}} = 164.52 \text{kips} \]

Since \( W_1 \leq \frac{420}{\sqrt{F_y}} \), \( V_m \leq \frac{2}{3} V_p = 109.68 \text{kips} \text{ Hence Ok} \)

Check against Moment Shear Interaction :

\[ R_1 = \frac{V_u}{\phi V_m} = 0.8653 \leq 1.0 \]
\[ R_2 = \frac{M_u}{\phi M_m} = 0.6907 \leq 1.0 \]

\[ R = \sqrt[3]{R_1^3 + R_2^3} = 0.9924 < 1.0 \text{ Hence Ok} \]
Calculation of length of Fillet Weld:

\[ A_f = b_f t_f = 3.4936 \text{ in}^2 \]

For reinforcing bars on one side of the web:

\[ A_r \leq \frac{A_f}{2} = 1.1645 \text{ in}^2 \text{ Hence OK} \]

\[ \frac{a_0}{h_0} = 2 \leq 2.5 \text{ Hence OK} \]

\[ V_1 = \frac{s_l}{t_w} = 14.25, \hspace{0.5cm} V_2 = \frac{s_b}{t_w} = 14.25 \]

\[ V_1 \text{ and } V_2 \leq \frac{140}{\sqrt{F_y}} = 23.333 \text{kips} \text{ Hence OK} \]

\[ \frac{M_u}{V_{ud}} = 3.4977 \leq 20 \text{ Hence OK} \]

\[ R_{w1} = \phi 2P_r = 3.375 \text{kips} \text{ (strength of weld within the length of the opening)} \]

\[ L_1 = \max \left( \frac{a_0}{4}, \frac{A_r \sqrt{3}}{2t_w} \right) = 5 \text{ in} \text{ (length extended on each side of the opening)} \]

Thus, Length of bar = \( a_0 + 2L_1 = 30 \) in

\[ R_{w2} = \phi F_y A_r = 1.6875 \text{kips} \text{ (strength of weld for extension on each side of opening)} \]

Strength of Weld \( R_{wr} = \max (R_{w1}, R_{w2}) = 3.375 \text{ kips} \)

Fillet Weld Size = 0.0044 in (rounded to nearest weld size of 0.0625 in = 1/16 in)

Corner Radii:

Minimum Radii = \( \max \left( 2t_w, \frac{5}{8} \right) = 0.76 \text{ in} \)

Comparison

Table 94: Web opening design for member no. 5

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Hand Calculations</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interaction Ratio, R</td>
<td>0.99</td>
<td>0.99</td>
<td>None</td>
</tr>
<tr>
<td>Reinforcement bar (required at web opening)</td>
<td>Length, in</td>
<td>30</td>
<td>30</td>
</tr>
<tr>
<td></td>
<td>Width, in</td>
<td>0.25</td>
<td>0.25</td>
</tr>
<tr>
<td></td>
<td>Thickness, in</td>
<td>0.1875</td>
<td>0.1875</td>
</tr>
<tr>
<td>Fillet Weld Size, in</td>
<td>1/16</td>
<td>0.0625 (1/16&quot;)</td>
<td>None</td>
</tr>
</tbody>
</table>

**STAAD Input**

STAAD PLANE EXAMPLE PROBLEM NO. 1

START JOB INFORMATION
ENGINEER DATE 18-May-05
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0; 3 0 20 0; 4 10 20 0; 5 20 20 0; 6 30 20 0; 7 0 35 0;
8 30 35 0; 9 7.5 35 0; 10 22.5 35 0; 11 15 35 0; 12 5 38 0; 13 25 38 0;
14 10 41 0; 15 20 41 0; 16 15 44 0;
MEMBER INCIDENCES
1 1 3; 2 3 7; 3 2 6; 4 6 8; 5 3 4; 6 4 5; 7 5 6; 8 7 12; 9 12 14;
10 14 16; 11 15 16; 12 13 15; 13 8 13; 14 9 12; 15 9 14; 16 11 14;
17 11 15; 18 10 15; 19 10 13; 20 7 9; 21 9 11; 22 10 11; 23 8 10;
MEMBER PROPERTY AMERICAN
1 3 4 TABLE ST W14X90
2 TABLE ST W10X49
5 6 7 TABLE ST W21X50
8 TO 13 TABLE ST W18X35
14 TO 23 TABLE ST L40404
MEMBER TRUSS
14 TO 23
MEMBER RELEASE
5 START MZ
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
DENSITY 0.000283
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
BETA 90 MEMB 3 4
MATERIAL MATERIAL1 MEMB 1 TO 4 6 TO 23
MATERIAL STEEL MEMB 5
UNIT FEET KIP
SUPPORTS
1 FIXED
2 PINNED
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTIES LIST 1 2 5 8 14
LOAD 1 DEAD AND LIVE LOAD
SELFWEIGHT X 1
SELFWEIGHT Y -1
JOINT LOAD
4 5 FY -15
11 FY -35
MEMBER LOAD
8 TO 13 UNI Y -0.9
6 UNI GY -1.2
CALCULATE RAYLEIGH FREQUENCY
LOAD 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 0.6
8 TO 10 UNI Y -1
`* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
LOAD COMB 4 75 PERCENT DL LL WL
1 2.75 2 2.75
PERFORM ANALYSIS
LOAD LIST 4
UNIT INCHES KIP
PARAMETER
CODE AISC
*WEB OPENINGS
*********************
RHOLE 0.6 MEMB 5
RDIM 20.0 10.0 MRMB 5
electrode 3
*********************
CHECK CODE MEMB 5 6
FINISH

STAAD Output

**Note:** A selected portion of the output file is shown here to demonstrate the design results.

```
STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)
===============================================
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE NOTED)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
            FX            MY             MZ       LOCATION
=======================================================================
*    5  ST   W21X50                   (AISC SECTIONS)
FAIL     AISC- H2-1         2.283         4
50.64 T          0.00       -4151.66      120.00

OUTPUT FOR WEB OPENING
----------------------
SECTION    LOAD        MZ/           FY/                IR
MOM. CAP. (Mm)  SHR. CAP. (Vm) ((MZ/Mm)^3+(FY/
Vm)^3)^0.33
=======================================================================
0.600       4        2510.67         34.46
3634.88         39.82             0.99

REINFORCING BARS : TO BE PLACED ON ONE SIDE OF WEB
BAR DIMENSION : LENGTH = 30.00 WIDTH = 0.25 THICKNESS = .1875
FILLET WELD : SIZE REQD = 0.0625 inch LENGTH REQD = 30.00

======================== END WEB OPENING OUTPUT =========================
*    6  ST   W21X50                   (AISC SECTIONS)
```
1B.3.6 Example 2: Deflection Check for a Steel Beam with Web Opening

This example illustrates the design procedure including a deflection check for a member with a web opening.

Problem

Design Method : ASD
Beam No : 5
Beam Section : W21X50

- \( d = 20.83 \text{ in} \)
- \( t_w = 0.38 \text{ in} \)
- \( b_f = 6.53 \text{ in} \)
- \( t_f = 0.535 \text{ in} \)
- \( Z = 110 \text{ in}^3 \)
- \( L = 30 \text{ ft} \)
- \( E = 29,732 \text{ ksi} \)

Opening Type: Rectangular

Number of openings : 1

- Section location of Opening, \( s_{open} = 0.3 \)
- \( a_0 = 20 \text{ in} \)
- \( h_0 = 10 \text{ in} \)
- \( e = 0 \text{ in} \) (concentric opening)
- \( F_y = 36 \text{ ksi} \)

Loading at critical location:

- \( V_u = 9.51 \text{ kip} \)
- \( M_u = 344.36 \text{ kip-inch} \)

Location of maximum deflection from start of the member : \( L_{critical} = 120 \text{ in} \)

Tee Properties:

- \( s_t = 5.415 \text{ in} \)
- \( s_b = 5.415 \text{ in} \)

\[
\text{Moment Area} = \frac{b_f t_f^2}{2} + t_w \left( s_t - t_f \right) \left( \frac{s_t t_f}{2} + t_f \right) = 6.4514 \text{in}^2
\]

\[
\text{Area} = b_f t_f + t_w (s_t - t_f) = 5.34795 \text{ in}^2
\]

\[
C_{yt} = \frac{\text{Moment Area}}{\text{Area}} = 1.20632 \text{ in.}
\]

\[
I_{zz,t} = I_{zz,b} = \frac{b_f t_f^3}{12} + b_f t_f \left( C_{yt} - \frac{t_f}{2} \right)^2 + t_w \left( \frac{t_f}{2} \right)^3 + t_w \left( s_t - t_f \right) \left( C_{yt} - t_f - \frac{s_t - t_f}{2} \right)^2 = 12.644 \text{in}^3
\]

(concentric opening)

\[
I_{zz} = I_{zz,t} + I_{zz,b} = 25.2872 \text{ in}^3
\]
Calculations

\[ R_t = \frac{I_{zz,b}}{I_{xx}} = 0.5 \]

\[ R_b = \frac{I_{zz,t}}{I_{xx}} = 0.5 \]

\[ V_t = F_y R_t = 4.755 \]

\[ V_b = F_y R_b = 4.755 \]

\[ \Delta_{0t} = \frac{V_t a_0^3}{6EI_{zz,t}} = 0.01686 \text{in} \]

\[ \Delta_{0b} = \frac{V_b a_0^3}{6EI_{zz,b}} = 0.01686 \text{in} \]

\[ \Delta_0 = \Delta_{0b} + \Delta_{0b} = 0.03373 \text{in} \]

\[ L_0 = \text{distance from high moment end of opening to adjacent support} = L - s_{open} - \frac{a_0}{2} = 314 \text{in} \]

\[ L_2 = \text{distance from support to the point where deflection is maximum} = L_0 \cdot L_{critical} = 240 \text{in} \]

\[ L_3 = L_0 \cdot L_2 = 74 \text{in} \]

\[ \Delta_{p1} = \frac{L_2}{L_0} \Delta_0 = 0.02578 \text{in} \]

\[ \theta_H = \frac{\Delta_0}{L_0} = 0.00011 \]

\[ \theta_T = \frac{V a_0^2}{4EI_{zz}} = 0.00126 \]

\[ \Delta_{p2} = \frac{2L_2 L_3 (\theta_H + \theta_T)}{L_0 + L_2} = 0.08799 \]

\[ \Delta = \Delta_{p1} + \Delta_{p2} = 0.11377 \text{in} \] (extra deflection due to opening)

Displacement factor: \( \Delta_{factor} = 750 \)

\[ \Delta_{allow} = \frac{L}{\Delta_{factor}} = 0.48 \text{in} \]

Ratio of \( \Delta_{actual}/\Delta_{allow} = 0.66 \) (without web opening)

Thus, \( \Delta_{actual} = 0.3168 \text{in} \) (from analysis considering stiffness of unperforated member without web opening)

Total deflection considering the effect of web hole,

\[ \Delta_{actual,wb} = \Delta + \Delta_{actual} = 0.4306 \text{in} \]

Ratio = \( \frac{\Delta_{actual,wb}}{\Delta_{allow}} = 0.897 \)

Comparison
Table 95: Web opening design for example 1

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Hand Calculations</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interaction Ratio, R</td>
<td>0.897</td>
<td>0.899</td>
<td>&lt; 1%</td>
</tr>
<tr>
<td>Total deflection, in</td>
<td>0.4306</td>
<td>0.4315*</td>
<td>&lt; 1%</td>
</tr>
</tbody>
</table>

* STAAD.Pro does not directly report the deflection, but rather the ratio of deflection with length, dff, for comparison with a limiting displacement factor.

\[
\Delta_{\text{actual,wb}} = \frac{L}{\text{dff}} = \frac{360 \text{ in}}{834.32} = 0.4315 \text{ in}
\]

**Note:** The member length used for deflection calculations is defined using the start deflection joint number, DJ1, and end deflection joint number, DJ2. These are assigned to the member in the design code parameters. STAAD.Pro uses this to calculate the physical member length rather than the segment length of the analytical member for deflection checks.

STAAD Input

```plaintext
STAAD PLANE EXAMPLE PROBLEM NO. 1
START JOB INFORMATION
ENGINEER DATE 18-May-05
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0; 3 0 20 0; 4 10 20 0; 5 20 20 0; 6 30 20 0; 7 0 35 0;
8 30 35 0; 9 7.5 35 0; 10 22.5 35 0; 11 15 35 0; 12 5 38 0; 13 25 38 0;
14 10 41 0; 15 20 41 0; 16 15 44 0;
MEMBER INCIDENCES
1 1 3; 2 3 7; 3 2 6; 4 10 20 0; 5 20 20 0; 6 30 20 0; 7 0 35 0;
8 30 35 0; 9 7.5 35 0; 10 22.5 35 0; 11 15 35 0; 12 5 38 0; 13 25 38 0;
14 10 41 0; 15 20 41 0; 16 15 44 0;
MEMBER PROPERTY AMERICAN
1 3 4 TABLE ST W14X90
2 TABLE ST W10X49
5 6 7 TABLE ST W21X50
8 TO 13 TABLE ST W18X35
14 TO 23 TABLE ST L40404
MEMBER TRUSS
14 TO 23
MEMBER RELEASE
5 START MZ
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
DENSITY 0.000283
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
```

**Design**

D. Design Codes
CONSTANTS
BETA 90 MEMB 3 4
MATERIAL MATERIAL1 MEMB 1 TO 4 6 TO 23
MATERIAL STEEL MEMB 5
UNIT FEET KIP
SUPPORTS
1 FIXED
2 PINNED
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTIES LIST 1 2 5 8 14
LOAD 1 DEAD AND LIVE LOAD
SELFWEIGHT X 1
SELFWEIGHT Y -1
JOINT LOAD
4 5 FY -15
11 FY -35
MEMBER LOAD
8 TO 13 UNI Y -0.9
6 UNI GY -1.2
CALCULATE RAYLEIGH FREQUENCY
LOAD 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 0.6
8 TO 10 UNI Y -1
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 3
UNIT INCHES KIP
PARAMETER
CODE AISC
TRACK 2 MEMB 5
DFF 750
DJ1 3 MEMB 5
DJ2 6 MEMB 5
*WEB OPENINGS
*********************
RHOLE 0.3 MEMB 5
RDIM 20.0 10.0 MEMB 5
*********************
CHECK CODE MEMB 5
FINISH

STAAD Output

Note: A selected portion of the output file is shown here to demonstrate the design results.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER 5</td>
<td>AISC SECTIONS</td>
</tr>
</tbody>
</table>
Design
D. Design Codes

<table>
<thead>
<tr>
<th>Parameter</th>
<th>IN KIP INCH</th>
<th>L3 L3 IN KIP INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>92.20</td>
<td>FA = 21.60</td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>14.67</td>
<td>fa = 0.97</td>
</tr>
<tr>
<td>UNL =</td>
<td>120.00</td>
<td>FCZ = 19.24</td>
</tr>
<tr>
<td>CB =</td>
<td>1.00</td>
<td>FTZ = 21.60</td>
</tr>
<tr>
<td>CMY =</td>
<td>0.85</td>
<td>FCY = 27.00</td>
</tr>
<tr>
<td>CMZ =</td>
<td>0.85</td>
<td>FTZ = 27.00</td>
</tr>
<tr>
<td>FYLD =</td>
<td>36.00</td>
<td>fbz = 0.00</td>
</tr>
<tr>
<td>NSF =</td>
<td>1.00</td>
<td>Fby = 0.00</td>
</tr>
<tr>
<td>DFF =</td>
<td>750.00</td>
<td>Fey = 18.01</td>
</tr>
<tr>
<td>dff=</td>
<td>834.32</td>
<td>Fez = 711.71</td>
</tr>
<tr>
<td>(KL/R)max =</td>
<td>92.20</td>
<td>(WITH LOAD NO.)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FV = 14.40</td>
</tr>
<tr>
<td></td>
<td></td>
<td>fv = 1.39</td>
</tr>
</tbody>
</table>

MAX FORCE/ MOMENT SUMMARY (KIP-FEET)

<table>
<thead>
<tr>
<th></th>
<th>VALUE</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOMENT-Y</th>
<th>MOMENT-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>AXIAL</td>
<td>-14.2</td>
<td>9.6</td>
<td>0.0</td>
<td>0.0</td>
<td>94.4</td>
</tr>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>10.0</td>
</tr>
<tr>
<td>LOADING</td>
<td>3</td>
<td>3</td>
<td>0</td>
<td>0</td>
<td>3</td>
</tr>
</tbody>
</table>

DESIGN SUMMARY (KIP-FEET)

<table>
<thead>
<tr>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>PASS</td>
<td>DEFLECTION</td>
<td>0.899</td>
<td>3</td>
</tr>
<tr>
<td>14.19 T</td>
<td>0.00</td>
<td>0.00</td>
<td>10.00</td>
</tr>
</tbody>
</table>

EXAMPLE PROBLEM NO. 1

--- PAGE NO. 7 ---

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
D1.C. American Codes - Steel Design per AISC LRFD Specification

The 2nd and 3rd editions of the American AISC LRFD code have been implemented. The commands to access those respective codes are:

For the 3rd edition code:

```plaintext
PARAMETER
CODE LRFD
```

or

```plaintext
PARAMETER
CODE LRFD3
```

For the 2nd Edition:

```plaintext
PARAMETER
CODE LRFD2
```

D1.C.1 General Comments

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design, the current state-of-the-art in structural engineering. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized—ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.
The following sections describe the salient features of the LRFD specifications as implemented in STAAD steel design. A detailed description of the design process along with its underlying concepts and assumptions is available in the LRFD manual. However, since the design philosophy is drastically different from the conventional Allowable Stress Design (ASD), a brief description of the fundamental concepts is presented here to initiate the user into the design process.

D1.C.2 LRFD Fundamentals

The primary objective of the LRFD Specification is to provide a uniform reliability for all steel structures under various loading conditions. This uniformity cannot be obtained with the allowable stress design (ASD) format. The ASD method can be represented by the inequality

\[ \Sigma Q_i < R_n/F.S. \]

The left side is the required strength, which is the summation of the load effects, \( Q_i \) (forces and moments). The right side, the design strength, is the nominal strength or resistance, \( R_n \), divided by a factor of safety. When divided by the appropriate section property (area or section modulus), the two sides of the inequality become the actual stress and allowable stress respectively. ASD, then, is characterized by the use of unfactored "working" loads in conjunction with a single factor of safety applied to the resistance. Because of the greater variability and, hence, unpredictability of the live load and other loads in comparison with the dead load, a uniform reliability is not possible.

LRFD, as its name implies, uses separate factors for each load and resistance. Because the different factors reflect the degree of uncertainty of different loads and combinations of loads and of the accuracy of predicted strength, a more uniform reliability is possible. The LRFD method may be summarized by the inequality

\[ y_i Q_i < R_n \Theta \]

On the left side of the inequality, the required strength is the summation of the various load effects, \( Q_i \), multiplied by their respective load factors, \( y_i \). The design strength, on the right side, is the nominal strength or resistance, \( R_n \), multiplied by a resistance factor, \( \Theta \).

In the STAAD implementation of LRFD, it is assumed that the user will use appropriate load factors and create the load combinations necessary for analysis. The design portion of the program will take into consideration the load effects (forces and moments) obtained from analysis. In calculation of resistances of various elements (beams, columns etc.), resistance (nominal strength) and applicable resistance factor will be automatically considered.

D1.C.3 Analysis Requirements

The types of construction recognized by AISC specification have not changed, except that both "simple framing" (formerly Type 2) and "semi-rigid framing" (formerly Type 3) have been combined into the same category, Type PR (partially restrained). "Rigid Framing" (formerly Type 1) is now Type FR (fully restrained). Type FR construction is permitted unconditionally. Type PR construction may necessitate some inelastic, but self-limiting, deformation of a structural steel element. Thus, when specifying Type PR construction, the designer should take into consideration the effects of partial restraint on the stability of the structure, lateral deflections and second order bending moments. As stated in Sect. C1 of the LRFD specification, an analysis of second order effects is required. Thus, when using LRFD code for steel design, the user must use the P-Delta analysis feature of STAAD.
D1.C.4 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, noncompact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and user specified shapes and design accordingly.

D1.C.5 Limit States

D1.C.5.1 Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (see D1.C.6 Design Parameters (on page 1454)). STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly.

D1.C.5.2 Axial Compression

The column strength equations have been revised in LRFD to take into account inelastic deformation and other recent research in column behavior. Two equations governing column strength are available, one for inelastic buckling and the other for elastic or Euler buckling. Both equations include the effects of residual stresses and initial out-of-straightness. Compression strength for a particular member is calculated by STAAD according to the procedure outlined in Chapter E of the LRFD specifications. For slender elements, the procedure described in Appendix B5.3 is used.

Singly symmetric and unsymmetric compression members are designed on the basis of the limit states of flexural-torsional and torsional buckling. The procedure of Appendix E3 is implemented for the determination of design strength for these limit states.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ and/or LY, LZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

D1.C.5.3 Flexural Design Strength

In LRFD, the flexural design strength of a member is determined by the limit state of lateral torsional buckling. Inelastic bending is allowed and the basic measure of flexural capacity is the plastic moment capacity of the section. The flexural resistance is a function of plastic moment capacity, actual laterally unbraced length, limiting laterally unbraced length, buckling moment and the bending coefficient. The limiting laterally unbraced length \( L_r \) and buckling moment \( M_r \) are functions of the section geometry and are calculated as per the procedure of Chapter F. The purpose of bending coefficient \( C_B \) is to account for the influence of the moment gradient on lateral-torsional buckling. This coefficient can be specified by the user through the use of parameter \( C_B \) (see D1.C.6 Design Parameters (on page 1454)) or may be calculated by the program (if \( C_B \) is specified as 0.0). In the absence of the parameter \( C_B \), a default value of 1.0 will be used. The procedure for calculation of design strength...
for flexure also accounts for the presence of residual stresses of rolling. To specify laterally unsupported length, either of the parameters UNB and UNT (see D1.C.6 Design Parameters (on page 1454)) can be used.

**D1.C.5.4 Combined Axial Force and Bending**

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas H1-1a and H1-1b. These interaction formulas cover the general case of biaxial bending combined with axial force. They are also valid for uniaxial bending and axial force.

**D1.C.5.5 Design for Shear**

The procedure of Sect. F2 of the LRFD Specification is used in STAAD to design for shear forces in members. Shear strength as calculated in LRFD is governed by the following limit states: Eq. F2-1a by yielding of the web; Eq. F2-2a by inelastic buckling of the web; Eq. F2-3a by elastic buckling of the web. Shear in wide flanges and channel sections is resisted by the area of the web, which is taken as the overall depth times the web thickness.

**D1.C.6 Design Parameters**

Design per LRFD specifications is requested by using the CODE parameter. Other applicable parameters are summarized in Table 2-4. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

The parameters DMAX and DMIN are applicable for member selection only.

**Table 96: AISC LRFD (2nd and 3rd Ed.) Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AXIS</td>
<td>1</td>
<td>1 = Design single angles for bending about their principle axis. 2 = Design single angles for bending about their geometric axis.</td>
</tr>
</tbody>
</table>
### Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = design at start and end nodes and those locations specified by the SECTION command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes.</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = deflection check based on the principle that maximum deflection is of the cantilever type (see D1.B.1.2 Design Parameters (on page 1409))</td>
</tr>
<tr>
<td>CB**</td>
<td>1.0</td>
<td>Coefficient Cb per Chapter F of AISC LRFD. If Cb is set to 0.0, it will be calculated by the program. Any of value will be used directly in design.</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of Member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot; (see D1.B.1.2 Design Parameters (on page 1409))</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of Member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot; (see D1.B.1.2 Design Parameters (on page 1409))</td>
</tr>
<tr>
<td>DMAX</td>
<td>45.0 in.</td>
<td>Maximum allowable section depth.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in.</td>
<td>Minimum allowable section depth</td>
</tr>
</tbody>
</table>

**Note:** See D1.A.6 Design Parameters (on page 1378).
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| FLX            | 1             | 1 = Single angle member is *not* fully braced against lateral torsional buckling.  
|                |               | 2 = Single angle member is fully braced against lateral torsional buckling.  
|                |               | 3 = Single angle member is braced against lateral torsional buckling at the point of maximum moment. |
| FYLD           | 36.0 ksi      | Yield strength of steel. |
| FU             | 60.0 ksi      | Ultimate tensile strength of steel. |
| KX             | 1.0           | K value for flexural-torsional buckling. |
| KY             | 1.0           | Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis. |
| KZ             | 1.0           | Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis. |
| LX             | Member Length | Length used for flexural-torsional buckling. |
| LY             | Member Length | Length to calculate the slenderness ratio for buckling about the local y-axis. |
| LZ             | Member Length | Length to calculate the slenderness ratio for buckling about the local z-axis. |
| MAIN           | 0.0           | 0.0 = check for slenderness  
|                |               | 1.0 = suppress slenderness check  
<p>|                |               | Any value greater than 1 = Allowable KL/r in compression. |
| NSF            | 1.0           | Net section factor for tension members. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROFILE</td>
<td></td>
<td>Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual load effect to design strength.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Member Length or depth, whichever is greater</td>
<td>Spacing of stiffeners for beams for shear design.</td>
</tr>
<tr>
<td>STP</td>
<td>1</td>
<td>Section type to determine $F_r$ (compressive residual stress in flange) per 3rd Ed. LRFD spec., p 16.1-97.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$1 = \text{Rolled section } (F_r = 10 \text{ ksi})$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$2 = \text{Welded section } (F_r = 16.5 \text{ ksi})$</td>
</tr>
<tr>
<td>TMAIN</td>
<td>300</td>
<td>Any value greater than 1 = Allowable KL/r in tension.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Specified the level of detail included in the output.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$0.0 = \text{Suppress all design strengths}$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$1.0 = \text{Print all design strengths}$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$2.0 = \text{Print expanded design output}$</td>
</tr>
<tr>
<td>UNB</td>
<td>Member Length</td>
<td>Unsupported length ($L_{un}$) of the bottom* flange for calculating flexural strength. Will be used only if flexural compression is on the bottom flange.</td>
</tr>
<tr>
<td>UNT</td>
<td>Member Length</td>
<td>Unsupported length ($L_{un}$) of the top* flange for calculating flexural strength. Will be used only if flexural compression is on the top flange.</td>
</tr>
</tbody>
</table>

*Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used).

**Note:** For a description of the deflection check parameters DFF, DJ1, DJ2, and CAN, see the D1.B.1.2 Design Parameters (on page 1409) of this manual.
The STIFF parameter represents the term “a” as defined in Section F2, page 6-113 of the LRFD 2nd edition manual.

** Non-default values of CB must be re-entered before every subsequent CHECK CODE or SELECT command.

**D1.C.7 Code Checking and Member Selection**

Both code checking and member selection options are available in STAAD LRFD implementation. See D1.B.1.3 Code Checking (on page 1418) and D1.B.1.4 Member Selection (on page 1419) for general information on these options.

Example for the LRFD-2001 code

```plaintext
UNIT KIP INCH
PARAMETER
CODE LRFD
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

Example for the LRFD-1994 code

```plaintext
UNIT KIP INCH
PARAMETER
CODE LRFD2
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

**Related Links**

- TR.49 Code Checking Specification (on page 2852)
- TR.49.1 Member Selection Specification (on page 2853)
- TR.49.2 Member Selection by Optimization (on page 2854)

**D1.C.8 Tabulated Results of Steel Design**

Results of code checking and member selection are presented in a tabular format. A detailed discussion of the format is provided in D1.B.1.9 Tabulated Results of Steel Design (on page 1421). Following exceptions may be noted: CRITICAL COND refers to the section of the LRFD specifications which governed the design.

If the TRACK is set to 1.0, member design strengths will be printed out.

**D1.C.9 Composite Beam Design per the AISC LRFD 3rd edition code**

The design of composite beams per the 3rd edition of the American LRFD code has been implemented. The salient points of this feature are as follows:
Table 97: Composite Beam Design Parameters for AISC-LRFD

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RBH</td>
<td>0.0 in.</td>
<td>Rib height for steel form deck.</td>
</tr>
<tr>
<td>EFFW</td>
<td>Value used in analysis</td>
<td>Effective width of the slab.</td>
</tr>
<tr>
<td>FPC</td>
<td>Value used in analysis</td>
<td>Ultimate compressive strength of the concrete slab.</td>
</tr>
</tbody>
</table>

**Theoretical Basis**

1. Find the maximum compressive force carried by concrete as:
   \[
   0.85 \cdot f_c \cdot b \cdot t
   \]
2. Find the maximum tensile force carried by the steel beam as:
   \[
   A_s \cdot f_y
   \]
   Tensile strength of concrete is ignored.
3. If step 1 produces a higher value than step 2, plastic neutral axis (PNA) is in the slab. Else, it is in the steel beam.

Location of the Plastic Neutral Axis (PNA) defines the moment capacity:

- Case 1: PNA in the slab
  Find the depth of the PNA below the top of the slab as:
$0.85f_c \cdot b \cdot a = A_s \cdot f_y$

Rearranging terms:

$$a = A_s \cdot f_y / (0.85f_c \cdot b)$$

![Diagram of Plastic neutral axis in the concrete slab](image)

**Figure 151: Plastic neutral axis in the concrete slab**

Lever arm

$$e = d/2 + h_r + t - a/2$$

Moment Capacity

$$\varphi_b (A_s \cdot f_y) e$$

- Case 2: PNA in Steel Beam

Define:

- $C_s = \text{Compressive force in slab} = 0.85f_c \cdot b \cdot t$
- $C_b = \text{Compressive force in steel beam}$
- $T_b = \text{Tensile force in steel beam}$

$$C_s + C_b = T_b$$

Since the magnitude of $C_b + \text{magnitude of} T_b = A_s \cdot f_y$

Substituting for $T_b$ as $(A_s \cdot f_y \cdot C_b)$ gives:

$$C_s + C_b = A_s \cdot f_y \cdot C_b$$

Rearranging terms:

$$C_b = (A_s \cdot f_y \cdot C_b) / 2$$

Determine whether the PNA is within the top flang of the steel beam or inside the web:

- $C_f = \text{Maximum compressive force carried by the flange} = A_f \cdot f_y$

Where:
If $C_f \geq C_b$, the PNA lies within the flange (Case 2A)
If $C_f < C_b$, the PNA lies within the web (Case 2B)

- **Case 2A: PNA in Flange of Steel Beam**

  Calculate:
  \[
y = C_f / (b_f \cdot f_y)
  \]

  Where:
  \[
b_f = \text{width of the flange}
  \]

  The point of action of the tensile force is the centroid of the steel are below the PNA. After find that point, $e_1$ and $e_2$ can be calculated.

**Figure 152: Plastic neutral axis falls within the top flange**

Moment Capacity

\[
\phi_b (C_f \cdot e_1 + C_s \cdot e_2)
\]

- **Case 2B: PNA in Web of Steel Beam**
$C_w = \text{Compressive force in the web} = C_b - C_f$

$g = \frac{C_w}{(t_w \cdot f_y)}$

Where:

$t_w = \text{thickness of the web}$

The point of action of the tensile force is the centroid of the steel area below the PNA. After finding that point, $e_1, e_2,$ and $e_3$ can be calculated.

Moment Capacity

$$\phi_b (C_s \cdot e_2 + C_f \cdot e_1 + C_w \cdot e_3)$$

Utilization Ratio = Applied Moment / Moment Capacity

**Notes**

1. Rib Height is the distance from top of flange of steel beam to lower surface of concrete.
2. If the slab is flush on top of the steel beam, set the Rib Height to zero.
Figure 154: Steel deck form ribs

3. For moments which cause tension in the slab (called positive moments in STAAD convention), design of the beam is presently not carried out.

4. Shear connectors are presently not designed.

5. Member selection is presently not carried out.

6. In order to design a member as a composite beam, the member property specification during the analysis phase of the data must contain the “CM” attribute. See TR.20.1 Assigning Properties from Steel Tables (on page 2461) for details.

```
Example

STAAD SPACE
...
MEMBER PROPERTY
1 TA CM W12X26 CT 6.0 FC 4.0 CW 40.0
...
PERFORM ANALYSIS
...
PARAMETER
CODE LRFD
RBH 5.0 MEMB 1
CHECK CODE MEMB 1
FINISH
```

D1.D. American Codes - Steel Design per AASHTO Specifications

Design to AASHTO Standard Specifications for Highway Bridges utilizing the ASD and LRFD approaches are available in STAAD.Pro. These are described in the following two sections.

To utilize the ASD method, specify the commands

```
PARAMETER
CODE AASHTO
```
or
PARAMETER
CODE AASHTO ASD
To utilize the LRFD method, specify the commands
PARAMETER
CODE AASHTO LRFD

D1.D.1 AASHTO (ASD)

The design of structural steel members in accordance with the AASHTO *Standard Specifications for Highway Bridges*, 17th edition has been implemented.

Related Links
• V. AASHTO 17th Ed ASD - Design Frame (on page 3828)

D1.D.1.1 General

The section of the above code implemented in STAAD is Chapter 10, Part C – Service Load design Method, Allowable Stress design. Sections 10.32.1.A and 10.36 are implemented. As per the AASHTO committee, this is the last edition for this code (the ASD approach) and only technical errors will be fixed in the future for this code.

In general, the concepts followed in MEMBER SELECTION and CODE CHECKING procedures are similar to that of the AISC based design. It is assumed that the user is familiar with the basic concepts of steel design facilities available in STAAD. Please refer to D. Steel Design (on page 944) of the STAAD Technical Reference Manual for detailed information on this topic. This section specifically addresses the implementation of steel design based on the AASHTO specifications.

Design is available for all standard sections listed in the AISC ASD 9th edition manual, namely, Wide Flanges, S, M, HP, Tees, Channels, Single Angles, Double Angles, Tubes and Pipes. The design of HSS sections (those listed in the 3rd edition AISC LRFD manual) and Composite beams (I shapes with concrete slab on top) are not supported.

D1.D.1.2 Allowable Stresses

The member design and code checking in STAAD.Pro is based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It is beyond the scope of this manual to describe every aspect of structural steel design per AASHTO specifications because of practical reasons. This section will discuss the salient features of the allowable stresses specified by the AASHTO code. Table 10.32.1A of the AASHTO code specifies the allowable stresses.

Axial Stress

 Allowable tension stress, as calculated in AASHTO is based on the net section. This tends to produce a slightly conservative result. Allowable tension stress on the net section is given by,

$$ F_t = 0.55 \cdot F_y $$

 Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$ F_a = F_y / (FS \cdot [1-(Kl/r)^2 F_y] / (4n^2E)) $$

when $(Kl/r) \leq Cc$
Fa = n²E/[FS·(Kl/r)²]

when (Kl/r) > Cc

Where:

C_c = (2π²E/F_y)¹/²

It should be noted that AASHTO does not have a provision for increase in allowable stresses for a secondary member and when 1/r exceeds a certain value.

Bending Stress

Allowable stress in bending compression for rolled shape girders and built-up sections whose compression flanges are supported laterally through their full length by embedment in concrete is given by:

F_b = 0.55·F_y

For similar members with unsupported or partially supported flange lengths, the allowable bending compressive stress is given by

F_b = 0.55·F_y[1 - (1/r)²F_y/(4π²E)]

Where:

r² = b²/12

Due to inadequate information in the AASHTO Code, the allowable tensile stresses due to bending for both axes are set to be the same as the corresponding allowable bending compressive stresses.

Shear Stress

Allowable shear stress on the gross section is given by:

F_v = 0.33 F_y

For shear on the web, the gross section is defined as the product of the total depth and the web thickness. The AASHTO code does not specify any allowable stress for shear on flanges. The program assumes the same allowable for shear stress (0.33F_y) for both shear on the web and shear on the flanges. For shear on the flanges, the gross section is taken as 2/3 times the total flange area.

Bending-Axial Stress Interaction

Members subjected to both axial and bending stresses are proportioned according to section 10.36 of the AASHTO steel code. All members subject to bending and axial compression are required to satisfy the following formula:

f_a/F_a + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0

at intermediate points, and

f_a/(0.472·F_y) + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0

at the ends of the member.

The start and end nodes of a member are treated as support points.

For members subject to axial tension and bending, the following equations are checked:

f_a/F_a + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0

at intermediate points, and

f_a/(0.472·F_y) + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0
at the ends of the member.

**D1.D.1.3 AASHTO (ASD) Design Parameters**

The following table outlines the parameters that can be used with the AASHTO (ASD) code along with the default values used if not explicitly specified.

**Table 98: AASHTO (ASD) Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = Design at ends and those locations specified by the SECTION command. 1.0 = Design at ends and every 1/12th point along the member length.</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Cb value as used in the calculation of Fb 0.0 = Cb value to be calculated Any other value will be used in the calculations.</td>
</tr>
<tr>
<td>CMY CMZ</td>
<td>0.85 for sidesway and calculated for no sidesway Cm value in local y &amp; z axes</td>
<td></td>
</tr>
<tr>
<td>DFF</td>
<td>None. (Mandatory for a deflection check)</td>
<td>“Deflection length” / Maximum allowable local axis deflection.</td>
</tr>
<tr>
<td>DJ1 DJ2</td>
<td>Start joint of member Joint No. denoting starting point for calculating “Deflection Length”.</td>
<td>End joint of member Joint No. denoting ending point for calculating “Deflection Length”.</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000.0</td>
<td>Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0</td>
<td>Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 KSI</td>
<td>Yield strength of steel in current units.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about local Y axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in local z-axis.</td>
</tr>
<tr>
<td>MAIN</td>
<td>0.0</td>
<td>0.0 = check for slenderness 1.0 = suppress slenderness check</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Ratio of &quot;Net cross section area&quot; / 'Gross section area' for tension member design.</td>
</tr>
<tr>
<td>PROFILE</td>
<td>None</td>
<td>Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>PUNCH</td>
<td></td>
<td>1.0 = K-Overlap 2.0 = K-Gap 3.0 = T and Y 4.0 = Cross with diaphragms 5.0 = Cross without diaphragms</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
</tr>
<tr>
<td>SSY</td>
<td>0.0</td>
<td>0.0 =Sidesway in local y-axis. 1.0 =No sidesway in local y-axis</td>
</tr>
<tr>
<td>SSZ</td>
<td>0.0</td>
<td>0.0 =Sidesway in local z-axis. 1.0 =No sidesway in local z-axis.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Greater of member length or depth of beam.</td>
<td>Spacing of stiffeners for plate girder design in current length units.</td>
</tr>
</tbody>
</table>
D1.D.2 AASHTO (LRFD)

The following outlines the implementation of the AASHTO Standard Specifications for Highway Bridges (LRFD, 1998) which has been implemented in STAAD.Pro.

Related Links
- V. AASHTO 2nd Ed LRFD - Design Beam (on page 3840)

D1.D.2.1 General

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design, the current state-of-the-art in structural engineering. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of AASTHO-LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the AASTHO-LRFD specifications as implemented in STAAD steel design.

D1.D.2.2 Capacities per AASHTO (LRFD) Code

Axial Strength

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| TRACK          | 0             | Level of detail in Output File:  
|                |               | 0 = Print the design output at the minimum detail level.  
|                |               | 1 = Print the design output at the intermediate detail level.  
|                |               | 2 = Print the design output at maximum detail level.  |
| UNF            | 1.0           | Unsupported length factor of the compression flange for calculating the allowable bending compressive strength.  |
| UNL            | Member Length | Unsupported length of compression flange for calculating allowable bending compressive stress.  |
| WSTR           | 0.4 x FYLD    | Allowable welding stress  |
The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified through the use of the parameter NSF. STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly.

\[
P_r = \varphi_y P_{ny} = \varphi_y F_y A_g\]

\[
P_r = \varphi_u P_{nu} = \varphi_u F_u A_n U\]

where

- \(P_{ny}\) = Nominal tensile resistance for yielding in gross section (kip)
- \(F_y\) = Yield strength (ksi)
- \(A_g\) = Gross cross-sectional area of the member (in\(^2\))
- \(P_{nu}\) = Nominal tensile resistance for the fracture in the net section (kip)
- \(F_u\) = Tensile strength (ksi)
- \(A_n\) = Net area of the member
- \(U\) = reduction factor to account for shear lag
- \(\varphi_y\) = resistance factor for yielding of tension member
- \(\varphi_u\) = resistance factor for fracture of tension members

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

\[
\lambda = \left(\frac{KL}{r_e \pi}\right)^2 \frac{F_y}{E}
\]

if \(\lambda \leq 2.25\)

Nominal compressive resistance,

\[
P_n = 0.66\lambda F_y A_s
\]

if \(\lambda > 2.25\)

Nominal compressive resistance

\[
P_n = 0.88 F_y A_s/\lambda
\]

where

- \(A_s\) = Gross sectional area

The Factored resistance

\[
P_r = \varphi_c P_n
\]

Bending Strength

The flow to calculate the allowable bending strength for rolled shape girders and built-up sections is given by the following flow chart.
Shear Strength

The nominal shear resistance of un-stiffened webs of homogeneous girders shall be calculated as.

If \( D/t_w \leq 2.46\sqrt{E/F_{yc}} \), then
\[
V_n = V_p = 0.58F_{yw}D_t
\]
If $2.46\sqrt{(E/F_y)} < D/t_w \leq 3.07\sqrt{(E/F_y)}$, then
\[ V_n = 1.48\ t_w\ 2\sqrt{(E-F_y)} \]

If $D/t_w > 3.07\sqrt{(E/F_y)}$, then
\[ V_n = 4.55\ t_w\ 3\ E/D \]

Bending-Axial Interaction

Members subjected to both axial forces and bending moments are proportioned according to section 6.9.2.2 of the AASHTO steel code. All members subject to bending and axial compression or axial tension are required to satisfy the following formula:

If $P_u/P_r < 0.2$, then
\[ \frac{P_u}{2.0 P_r} + \left( \frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \leq 1.0 \]

If $P_u/P_r \geq 0.2$, then
\[ \frac{P_u}{P_r} + \frac{8}{9} \left( \frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \leq 1.0 \]

D1.D.2.3 AASHTO (LRFD) Design Parameters

The following table outlines the parameters that can be used with the AASHTO (LRFD) code along with the default values used if not explicitly specified.

**Table 99: AASHTO (LRFD) Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| BEAM           | 1.0           | Identify where beam checks are performed:  
0 = Perform design at ends and those locations specified in the SECTION command.  
1 = Perform design at ends and 1/12th section locations along member length. |
<p>| DMAX           | 1000          | Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command. |
| DMIN           | 0             | Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DFF</td>
<td>0</td>
<td>&quot;Deflection Length&quot;/Max allowable local deflection. If set to 0, (default) then no deflection check is performed.</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start joint of member</td>
<td>Joint No. denoting starting point for calculating &quot;Deflection Length&quot;.</td>
</tr>
<tr>
<td>DJ2</td>
<td>End joint of member</td>
<td>Joint No. denoting ending point for calculating &quot;Deflection Length&quot;.</td>
</tr>
<tr>
<td>GRADE</td>
<td>1</td>
<td>Grade of Steel:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1: Grade 36</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2: Grade 50</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 Grade 50W</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4: Grade 70W</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5: Grade 100/100W</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refer to AASHTO LRFD, Table 6.4.1-1</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about the local Y axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in local z-axis.</td>
</tr>
<tr>
<td>MAIN</td>
<td>0.0</td>
<td>Flag for checking slenderness limit:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0 = check for slenderness</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = suppress slenderness check</td>
</tr>
</tbody>
</table>
### Parameter Name | Default Value | Description
--- | --- | ---
NSF | 1.0 | Net Section Factor. Ratio of (Net Area)/(Gross Area)
NSF | 1.0 | Net section factor for tension members.
TRACK | 0 | Level of detail in Output File:
 | | 0 = Print the design output at the minimum detail level.
 | | 1 = Print the design output at the intermediate detail level.
 | | 2 = Print the design output at maximum detail level.
UNB | Member Length | Unsupported length of bottom flange. Used for calculating the moment of resistance when the bottom of beam is in compression.
UNT | Member Length | Unsupported length of top flange. Used for calculating the moment of resistance when top of beam is in compression.

---

**D1.E. American Codes - Steel Design per AISI Cold Formed Steel Code**

Provisions of the AISI Specification for the Design of Cold-Formed Steel Structural Members, 1996 Edition have been implemented. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations using the LRFD Method. For flexural members, the Nominal Section Strength is calculated on the basis of initiation of yielding in the effective section (Procedure I). Strength increase from Cold Work of Forming is a user selectable option.

**D1.E.1 Cross-Sectional Properties**

you specifies the geometry of the cross-section by choosing one of the section shape designations from the STAAD Steel Tables for cold-formed sections, which mirror the Gross Section Property Tables published in the "Cold-Formed Steel Design Manual", AISI, 1996 Edition.

The Tables are currently available for the following shapes:

- Channel with Lips
- Channel without Lips
- Angle with Lips
- Angle without Lips
- Z with Lips
- Z without Lips
- Hat
Shape selection may be done using the member property pages of the graphical user interface (GUI) or by specifying the section designation symbol in the input file. Details of the latter are explained below.

### D1.E.2 The AISI Steel Section Library

The command-line syntax for assigning steel sections from the AISI library is as explained below:

<table>
<thead>
<tr>
<th>Section Type</th>
<th>Section ID</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>C-Section with Lips</td>
<td>20 TO 30 TA ST</td>
<td>14CS3.75X135</td>
</tr>
<tr>
<td>C-Section with Lips</td>
<td>33 36 TA ST</td>
<td>12CS1.625X102</td>
</tr>
<tr>
<td>C-Section with Lips</td>
<td>42 43 TA ST</td>
<td>4CS4X060</td>
</tr>
<tr>
<td>C-Section with Lips</td>
<td>50 TO 60 TA ST</td>
<td>10 CU1.25X071</td>
</tr>
<tr>
<td>C-Section with Lips</td>
<td>32 33 TA ST</td>
<td>3CU1.25X057</td>
</tr>
<tr>
<td>C-Section with Lips</td>
<td>21 28 TA ST</td>
<td>1.5CU1.25X035</td>
</tr>
<tr>
<td>Z-Section with Lips</td>
<td>1 3 4 TA ST</td>
<td>12ZS3.25X135</td>
</tr>
<tr>
<td>Z-Section with Lips</td>
<td>33 45 TA ST</td>
<td>10ZS3X060</td>
</tr>
<tr>
<td>Z-Section with Lips</td>
<td>12 13 TA ST</td>
<td>6ZS2X048</td>
</tr>
<tr>
<td>Z-Section without Lips</td>
<td>2 3 TA ST</td>
<td>12U1.25X105</td>
</tr>
<tr>
<td>Z-Section without Lips</td>
<td>4 5 TA ST</td>
<td>4U1.25X036</td>
</tr>
<tr>
<td>Z-Section without Lips</td>
<td>6 7 TA ST</td>
<td>1.5U1.25X048</td>
</tr>
<tr>
<td>Equal Leg Angles without Lips</td>
<td>8 9 TA ST</td>
<td>4LS4X105</td>
</tr>
<tr>
<td>Equal Leg Angles without Lips</td>
<td>10 11 TA ST</td>
<td>3LS3X060</td>
</tr>
<tr>
<td>Equal Leg Angles without Lips</td>
<td>12 13 TA ST</td>
<td>2LS2X075</td>
</tr>
<tr>
<td>Equal Leg Angles without Lips</td>
<td>1 5 TA ST</td>
<td>4LU4X135</td>
</tr>
<tr>
<td>Equal Leg Angles without Lips</td>
<td>7 8 TA ST</td>
<td>2.5LU2.5X105</td>
</tr>
<tr>
<td>Equal Leg Angles without Lips</td>
<td>4 9 TA ST</td>
<td>2LU2X060</td>
</tr>
<tr>
<td>Hat Sections without Lips</td>
<td>4 8 TA ST</td>
<td>10HU5X075</td>
</tr>
<tr>
<td>Hat Sections without Lips</td>
<td>5 6 TA ST</td>
<td>6HU9X105</td>
</tr>
<tr>
<td>Hat Sections without Lips</td>
<td>1 7 TA ST</td>
<td>3HU4.5X135</td>
</tr>
</tbody>
</table>

### D1.E.3 Current Limitations

At the present time, only standard single sections are available for specification. Options such as double angles, double channels, and user provided sections including pipes and tubes will be available at a later date. Additionally, combination sections, such as an angle placed on top of a channel, or a plate welded to the top, bottom or side of one of the above shapes, are not available at this time.

STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.
D1.E.4 Design Procedure

The following two design modes are available:

1. Code Checking

   The program compares the resistance of members with the applied load effects, in accordance with the LRFD Method of the AISI code. Code checking is carried out for locations specified by the user via the SECTION command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. The user may choose the degree of detail in the output data by setting the TRACK parameter.

2. Member Selection

   You may request that the program search the cold formed steel shapes database (AISI standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, present design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

The program calculates effective section properties in accordance with the following Sections:

- B2.1, Uniformly Compressed Stiffened Elements
- B2.3, Webs and Stiffened Elements with Stress Gradient
- B3.1, Uniformly Compressed Unstiffened Elements
- B3.2, Unstiffened Elements and Edge Stiffeners with Stress Gradient
- B4.2, Uniformly Compressed Elements with an Edge Stiffener

Cross-sectional properties of members are checked for compliance with the following Sections:

- B1.1(a), Maximum Flat-Width-to-Thickness Ratios, and
- B1.2, Maximum Web Depth-to-Thickness Ratio

The program checks member strength in accordance with Chapter C of the specification as follows:

1. Tension Members

   Resistance is calculated in accordance with Section C2.

2. Flexural Members

   a. C3.1, Strength for bending only:

      - C3.1.1, Nominal Section Strength, Procedure I
      - C3.1.2, Lateral Buckling Strength

   b. C3.2, Strength for Shear Only

   c. C3.3, Strength for Combined Bending and Shear

3. Concentrically Loaded Compression Members.

   a. C4.1, Sections not subject to Torsional or Torsional-Flexural Buckling, and
   b. C4.2, Doubly or Singly Symmetric sections subject to Torsional or Torsional-Flexural Buckling.

4. Combined Axial Load and Bending.

   a. C5.1, Combined Tensile Axial Load and Bending, and
   b. C5.2, Combined Compressive Axial Load and Bending.
## D1.E.5 Design Parameters

The following table contains the input parameters for specifying values of design variables and selection of design options.

**Table 100: AISI Cold Formed Steel Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = design at start and end nodes and those locations specified by the SECTION command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes.</td>
</tr>
<tr>
<td>CMZ</td>
<td>1.0</td>
<td>End moment coefficient for bending about z-axis. See AISI C5.2.2. Used for combined axial load and bending design. Values range from 0.4 to 1.0.</td>
</tr>
<tr>
<td>CMY</td>
<td>0.0</td>
<td>End moment coefficient for bending about y-axis. See AISI C5.2.2. Used for combined axial load and bending design. Values range from 0.4 to 1.0.</td>
</tr>
<tr>
<td>CWY</td>
<td>0</td>
<td>Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See AISI A7.2. 0 = effect not included 1 = effect included</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000.0</td>
<td>Maximum depth permissible for the section during member selection, in current units.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0</td>
<td>Minimum depth required for the section during member selection, in current units.</td>
</tr>
</tbody>
</table>

*Note: See [D1.A.6 Design Parameters](on page 1378).*
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| FLX            | 1             | Specifies whether torsional-flexural buckling restraint is provided or is not necessary for the member. See AISI C4.1  
0 = Section subject to torsional-flexural buckling and restraint not provided  
1 = Restraint provided or unnecessary |
<p>| FU             | 58 ksi        | Ultimate tensile strength of steel in current units. |
| FYLD           | 36 ksi        | Yield strength of steel in current units. |
| KT             | 1.0           | Effective length factor for torsional buckling used to compute the (KL/r) ratio for determining the capacity in axial compression. Values range from 0.01 (for a column completely restrained against torsional buckling) to any large value. |
| KY             | 1.0           | Effective length factor for overall column buckling about the local (y)-axis; used to compute the (KL/r) ratio for determining the capacity in axial compression. Values can range from 0.01 (for a column completely restrained against buckling) to any large value. |
| KZ             | 1.0           | Effective length factor for overall column buckling about the local (z)-axis; used to compute the (KL/r) ratio for determining the capacity in axial compression. Values can range from 0.01 (for a column completely restrained against buckling) to any large value. |
| LT             | Member Length | Unbraced length used in computing (KL/r) for twisting, in current units of length. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length used to calculate slenderness ratio for buckling about the local y-axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as LY, but in the local z-axis.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members. See AISI C2.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Member Length</td>
<td>Spacing in the longitudinal direction of shear stiffeners for reinforced web elements, in current units of length. See AISI C3.2.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>Used to control the level of detail in which the design output is reported in the output file.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 = Prints only the member number, section name, ration, and PASS/FAIL status.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Prints the design summary in addition to that printed by TRACK 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = Prints member and material properties in addition to that printed by TRACK 1</td>
</tr>
<tr>
<td>TSA</td>
<td>1</td>
<td>Specifies whether the bearing and intermediate transverse stiffeners are present. If set to 1, the program uses more liberal set of interaction equations found in AISI C3.3.2.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 = Beams with unreinforced webs</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Beams with transverse web stiffeners</td>
</tr>
</tbody>
</table>

**D1.F. American Codes - Concrete Design per ACI 318**

Concrete member and floor design per ACI 318-14, ACI 318-11, ACI 318-08, ACI 318-05 ACI 318-02 and ACI 318-99 *Building Code Requirements for Structural Concrete* is available in STAAD.Pro.

**Note:** To use the ACI 318 design code for a specific publication year, refer to Design Operations.
D1.F.1 Design Operations

STAAD.Pro has the capabilities for performing concrete design. It will calculate the reinforcement needed for the specified concrete section. All the concrete design calculations are based on the current edition of ACI 318 (unless a previous version is specified).

The design requirements need to be contained within a concrete design block, include the code reference, any modifications to design parameters and instructions as to which objects are to be designed.

```
START CONCRETE DESIGN
CODE code_ref
parameters values
design_instructions
END CONCRETE DESIGN
```

The versions of the ACI 318 code implemented can be referred to as follows:

<table>
<thead>
<tr>
<th>Code Edition</th>
<th>code_ref</th>
<th>Design Beams</th>
<th>Design Columns</th>
<th>Design Slabs</th>
</tr>
</thead>
<tbody>
<tr>
<td>2014</td>
<td>ACI</td>
<td>Yes</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td></td>
<td>ACI 2014</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2011</td>
<td>ACI 2011</td>
<td>Yes</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td>2008</td>
<td>ACI 2008</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>2005</td>
<td>ACI 2005</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>2002</td>
<td>ACI 2002</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>1999</td>
<td>ACI 1999</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
</tbody>
</table>

For a list of the parameters that can be used for design, see D1.F.3 Design Parameters (on page 1481).

Note: Not all parameters apply to all versions of the ACI 318 design code.

```
design_instructions:
DESIGN BEAM { member_list | ALL }
DESIGN COLUMN { member_list | ALL }
DESIGN PLATE { element_list | ALL }
```

See also TR.53 Concrete Design Specifications (on page 2859).

Example

An partial input with an example of beam design input for ACI 318-14:

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2014
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
```
D1.F.2 Section Types for Concrete Design

The current version of ACI 318 in STAAD.Pro supports the design of beams and columns defined as follows.

![Rectangular, Circular, Tee Sections](image)

*Figure 156: Section types for concrete design*

Beam Design – uses a 2D beam element if specified with the following properties:

1. Rectangular
   - Defined as a Prismatic section with YD (depth) and ZD (width)
   - Defined in a User Table, type: prismatic

2. Tee Section
   - Defined as a Prismatic section with YD (overall depth), YB (depth of stem), ZD (width of flange), and ZB (width of stem)

Column Design – uses a 2D beam element if specified with the following properties:

1. Rectangular
   - Defined as a Prismatic section with YD (depth) and ZD (width)
   - Defined in a User Table, type: prismatic

2. Tee Section
   - Defined as a Prismatic section with YD (overall depth), YB (depth of stem), ZD (width of flange), and ZB (width of stem)

3. Circular
   - Defined as Prismatic section with YD (circumference)

Related Links

- TR.20.2 Prismatic Property Specification (on page 2465)

**D1.F.2.1 Section Types Supported for ACI 318-99 – ACI 318-11**

Design using ACI 318-11, ACI 318-08, ACI 318-05, ACI 318-02, or ACI 318-99 in STAAD.Pro supports the design of beams, columns, or slabs defined as follows.
Beam Design – uses a 2D beam element if specified with the following properties:

1. Rectangular
   a. Defined as a Prismatic section with YD (depth) and ZD (width)
   b. Defined in a User Table, type: prismatic
2. Tee Section
   a. Defined as a Prismatic section with YD (overall depth), YB (depth of stem), ZD (width of flange), and ZB (width of stem)
3. Trapezoidal
   a. Defined as Prismatic section with YD (depth), ZD (width at top), and ZB (width at bottom).

Column Design – uses a 2D beam element if specified with the following properties:

1. Rectangular
   a. Defined as a Prismatic section with YD (depth) and ZD (width)
   b. Defined in a User Table, type: prismatic
2. Circular
   a. Defined as Prismatic section with YD (circumference)

Floor Slab Design - uses a 2D plate element

1. 3-noded plate with thickness of element is equal to the thickness of the slab
2. 4-noded plate with thickness of element is equal to the thickness of the slab. All four nodes should be co-planar.

D1.F.3 Design Parameters

The following outlines the parameters that are available to control the design of members as beams or columns per the ACI 318-14 code. The commands that initiate the design of these is outlined in TR.53.3 Concrete Design Command (on page 2860). If any parameter is not specified and its value is required in the design, then it will use that specified as the default value in the tables below.

Note: Not all the parameters are used in all versions of the ACI 318 code. Ensure that all the necessary parameters for the version of the code are properly specified. Where practical, parameters used in earlier versions of the code will be supported in the newer versions. Some old parameters are not supported in newer versions of the code and new parameters have been added in newer versions of the code that provide greater flexibility in the design.
Table 101: ACI 318 Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| BDY            | 1.0           | Column stiffness reduction factor for bending about the local Y axis.  
**Note:** Not used if parameter MMSG > 0  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| BDZ            | 1.0           | Column stiffness reduction factor for bending about the local Z axis.  
**Note:** Not used if parameter MMSG > 0  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| BRDO           | 0             | Bottom Rebar Detail Options  
0) Increase number of bars, then increase bar size, then add layers  
1) Increase the bar size, then increase the number of bars, then add layers  
**Note:** Only used with ACI 318-14. |
| CLB            | 1.5 in. (40 mm) for members* | Minimum cover from the base of the section to the bottom longitudinal reinforcement. |
| CLS            | 1.5 in. (40 mm)* | Minimum cover from the sides of the section to the side longitudinal reinforcement.  
See **Note a** (on page 1486) and **Note c** (on page 1486) below. |
| CLT            | 1.5 in. (40 mm) for members* | Minimum cover from the top of the section to the top longitudinal reinforcement. |
| CRDO           | 0             | Design options for main reinforcement in columns  
0) Increase the number of bars and the bar size  
1) Increase the bar size and then the number of bars  
**Note:** Only used with ACI 318-14. |
<p>| FC             | 4,000 psi (28 N/mm²)* | Compressive strength of concrete. |
| FYMAIN         | 60,000 psi (420 N/mm²)* | Yield stress for main longitudinal reinforcing steel. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FYSEC</td>
<td>60,000 psi (420 N/mm²)*</td>
<td>Yield stress for transverse reinforcing steel.</td>
</tr>
<tr>
<td>LATR</td>
<td>0 (No)</td>
<td>Is this member subject to lateral loading?</td>
</tr>
<tr>
<td></td>
<td>0) No</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1) Yes</td>
<td></td>
</tr>
<tr>
<td></td>
<td>This will be used in determining</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the development length requirements,</td>
<td></td>
</tr>
<tr>
<td></td>
<td>in ACI 318-14 clause 9.7.3.8.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>This parameter is only applicable</td>
<td></td>
</tr>
<tr>
<td></td>
<td>for beams.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-14.</td>
<td></td>
</tr>
<tr>
<td>LWF</td>
<td>0</td>
<td>Specifies the aggregate composition of the concrete.</td>
</tr>
<tr>
<td></td>
<td>0) Normal weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1) Sand light weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2) All light weight</td>
<td></td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>#18 (Imperial), 57M (Metric)</td>
<td>Maximum size of reinforcing bar to be used for longitudinal reinforcement.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> When using ACI 318-14,</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the maximum size used in specific</td>
<td></td>
</tr>
<tr>
<td></td>
<td>positions can be individually set.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>See detailing parameters below.</td>
<td></td>
</tr>
<tr>
<td>MIMB</td>
<td>#4 (Imperial), 13M (Metric)</td>
<td>Minimum main rebar number (size) at the bottom face</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-14.</td>
<td></td>
</tr>
<tr>
<td>MIMS</td>
<td>#4 (Imperial), 13M (Metric)</td>
<td>Minimum main rebar number (size) at for skin (side) bars</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-14.</td>
<td></td>
</tr>
<tr>
<td>MIMT</td>
<td>#4 (Imperial), 13M (Metric)</td>
<td>Minimum main rebar number (size) at the top face</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-14.</td>
<td></td>
</tr>
<tr>
<td>MINMAIN</td>
<td>#4 (Imperial), 13M (Metric)</td>
<td>Minimum size of reinforcing bar to be used for longitudinal reinforcement.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> When using ACI 318-14,</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the minimum size used in specific</td>
<td></td>
</tr>
<tr>
<td></td>
<td>positions can be individually set.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>See detailing parameters below.</td>
<td></td>
</tr>
<tr>
<td>MINSEC</td>
<td>#4 (Imperial), 13M (Metric)</td>
<td>Minimum size of reinforcing bar to be used for transverse reinforcement.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| MMAG           | 1.0           | Factor to magnify the analysis moments used for column design.  
**Note:** For ACI 318-14, if this is set to 0, then the moments for the local Y and Z axis will be calculated independently using the parameters BDY and BDZ. |
| MXMB           | #18 (Imperial), 57M (Metric) | Maximum main rebar number (size) at the bottom face  
**Note:** Only used with ACI 318-14. |
| MXMS           | #18 (Imperial), 57M (Metric) | Maximum main rebar number (size) for skin (side faces) bars  
**Note:** Only used with ACI 318-14. |
| MXMT           | #18 (Imperial), 57M (Metric) | Maximum main rebar number (size) at the top face  
**Note:** Only used with ACI 318-14. |
| RCOAT          | 0             | Rebar coating type:  
0) No coating  
1) Epoxy coating  
2) Galvanized  
**Note:** Only used with ACI 318-14. |
| REINF          | 0             | Column transverse reinforcement type.  
0) links (tied column)  
1) spiral reinforcement |
| RHOMIN         | 0.01          | Minimum reinforcement required in a concrete column. Enter a value between 0.0 and 0.08, where 0.08 = 8% reinforcement; the maximum allowed by the ACI code.  
**Note:** This parameter is not used with ACI 318-14. |
| SKY            | 1.0           | Effective length factor for moment magnification procedure calculations for columns, in the local Y direction.  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| SKZ            | 1.0           | Effective length factor for moment magnification procedure calculations for columns, in the local Z direction.  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SLY</td>
<td>n/a</td>
<td>Number of the load case that applies sidesway load in the local Y axis for column design. The moments from this case are used to magnify the local Y axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the SWY parameter setting.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> This parameter is not used if MMAG has been specified.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-11 and ACI 318-14.</td>
</tr>
<tr>
<td>SLZ</td>
<td>n/a</td>
<td>Number of the load case that applies sidesway load in the local z axis for column design. The moments from this case are used to magnify the local Z axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the SWY parameter setting.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> This parameter is not used if MMAG has been specified.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-11 and ACI 318-14.</td>
</tr>
<tr>
<td>SQY</td>
<td>0.2</td>
<td>Stability index, Q, in the local Y axis as defined in ACI 318 section 6.6.4.3(b).</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-11 and ACI 318-14.</td>
</tr>
<tr>
<td>SQZ</td>
<td>0.2</td>
<td>Stability index, Q, in the local Z axis as defined in ACI 318 section 6.6.4.3(b).</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-11 and ACI 318-14.</td>
</tr>
<tr>
<td>SWY</td>
<td>0</td>
<td>Braced against sidesway:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0) Not braced</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1) Braced</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Use only when MMAG parameter is equal 0.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-11 and ACI 318-14.</td>
</tr>
</tbody>
</table>
### Parameter Name | Default Value | Description
---|---|---
**TRACK** | 0.0 | Output detail
| | | 0) Design and report only the steel area requirements
| | | 1) Perform detailed design and report bar layouts
| | | 2) Perform detailed design and report all details
**TRDO** | 0 | Top Rebar Detail Options
| | | 0) Increase number of bars, then increase the bar size, then add layers
| | | 1) Increase the bar size, then increase the number of bars, then add layers
**TRN** | 0 | Specifies whether a member is subject to transverse loads between supports. This parameter influences the moment magnification calculations per CL 6.6.4.5.3.
| | | 0) True
| | | 1) False

**Notes**

a. The value used when specifying the CLS parameter for column design is taken to be the clear cover for the longitudinal bars in a column. It is not taken as the clear cover for the tie bars. Therefore, the distance from the edge of the column to the centerline of the first row of longitudinal bars is CLS plus half the diameter of the main bar.

b. The following parameters are *not* applicable to the ACI 318-14 (only to ACI 318-11 and earlier):
   - DEPTH
   - EFACE
   - RHOMAIN
   - SFACE
   - WIDTH
   - NSECTION

   **Note:** In ACI 318-14, members are always designed with five sections. The parameter NSECTION used with older editions of the code has no effect.

c. MAXMAIN and MINMAIN are now supplemented by new set of parameters like MXMB, MIMC etc. You can assign values either in the old parameters (like MAXMAIN) or in the new parameters (like MXMT, MXMB, MXMC). If you choose to use older parameters like MAXMAIN and MINMAIN, the program will store this input into newer parameters like MXMB, MIMC etc. internally. However, if you use both older and newer parameters, only the newer parameter values are accepted by the program.

**Related Links**
- **TR.31.2.14 IBC 2015 Seismic Load Definition**
- **TR.53.2 Concrete Design-Parameter Specification**
D1.F.3.1 ACI 318-2011 Design Parameters

The following outlines the parameters that are available to control the design of members as beams or columns per the ACI 318-11 code. The commands that initiate the design of these is outlined in TR.53.3 Concrete Design Command (on page 2860). If any parameter is not specified and its value is required in the design, then it will use that specified as the default value in the tables below.

**Note:** Not all the parameters are used in all versions of the ACI 318 code. Ensure that all the necessary parameters for the version of the code are properly specified. Where practical, parameters used in earlier versions of the code will be supported in the newer versions. Some old parameters are not supported in newer versions of the code and new parameters have been added in newer versions of the code that provide greater flexibility in the design.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BDY</td>
<td>1.0</td>
<td>Column stiffness reduction factor for bending about the local Y axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Not used if parameter MMAG &gt; 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-11 and ACI 318-14.</td>
</tr>
<tr>
<td>BDZ</td>
<td>1.0</td>
<td>Column stiffness reduction factor for bending about the local Z axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Not used if parameter MMAG &gt; 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Only used with ACI 318-11 and ACI 318-14.</td>
</tr>
<tr>
<td>CLB</td>
<td>1.5 in. (40 mm) for beams* &lt;br&gt;0.75 in. (20 mm) for plate elements*</td>
<td>Minimum cover from the base of the section to the bottom longitudinal reinforcement.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>See <strong>Note c</strong> (on page 1491) below.</td>
</tr>
<tr>
<td>CLS</td>
<td>1.5 in. (40 mm)*</td>
<td>Minimum cover from the sides of the section to the side longitudinal reinforcement.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>See <strong>Note a</strong> (on page 1486) and <strong>Note c</strong> (on page 1491) below.</td>
</tr>
<tr>
<td>CLT</td>
<td>1.5 in. (40 mm) for beams* &lt;br&gt;0.75 in. (20 mm) for plate elements*</td>
<td>Minimum cover from the top of the section to the top longitudinal reinforcement.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>See <strong>Note c</strong> (on page 1491) below.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD*</td>
<td>Depth of the member measured in the local Y axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> This parameter is not used with ACI 318-14.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| EFACE          | 0.0*         | Face of support location at end of beam. If specified, the shear force at end is computed at a distance of EFACE + d from the end joint of the member.  
**Note:** See [D.1.F.4.3 Shear and Torsion Design](on page 1497) for additional information.  
**Note:** This parameter is not used with ACI 318-14. |
| FC             | 4,000 psi (28 N/mm²)* | Compressive strength of concrete. |
| FYMAIN         | 60,000 psi (420 N/mm²)* | Yield stress for main longitudinal reinforcing steel. |
| FYSEC          | 60,000 psi (420 N/mm²)* | Yield stress for transverse reinforcing steel. |
| LWF            | 1.0          | Modification factor, λ, for lightweight concrete as specified in ACI i318-08, cl. 8.6.1. Valid entries are between 0.75 and 1.0, inclusive. Used as a reduction factor for the mechanical properties of lightweight concrete.  
**Note:** This parameter is used in ACI 318-2008, 2001, and 2014 only. |
| MAXMAIN        | #18 bar (57 mm) | Maximum size of reinforcing bar to be used for longitudinal reinforcement.  
**Note:** When using ACI 318-14, the maximum size used in specific positions can be individually set. See detailing parameters below.  
**Note:** See [Note b](on page 1491) below. |
| MINMAIN        | #4 bar (13 mm) | Minimum size of reinforcing bar to be used for longitudinal reinforcement.  
**Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below.  
**Note:** See [Note b](on page 1491) below. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| MINSEC         | #4 bar (13 mm) | Minimum size of reinforcing bar to be used for transverse reinforcement.  
**Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below.  
**Note:** See Note b (on page 1491) below. |
| MMAG           | 1.0           | Factor to magnify the analysis moments used for column design.  
**Note:** For ACI 318-14, if this is set to 0, then the moments for the local Y and Z axis will be calculated independently using the parameters BDY and BDZ. |
| NSECTION       | 12            | Number of equally spaced sections to be considered in finding critical moments for beam design. The valid range is from 12 to 20.  
This parameter applies to the design process rather than individual members and thus should not have any assigned member list. If more than one NSECTION parameter is defined, then highest value will be used. |
| REINF          | 0             | Column transverse reinforcement type.  
0) links (tied column)  
1) spiral reinforcement |
| RHOMIN         | 0.01          | Minimum reinforcement required in a concrete column. Enter a value between 0.0 and 0.08, where 0.08 = 8% reinforcement; the maximum allowed by the ACI code.  
**Note:** This parameter is not used with ACI 318-14. |
| SFACE          | 0.0*          | Face of support location at start of beam. If specified, the shear force at start is computed at a distance of SFACE + d from the start joint of the member.  
**Note:** See D1.F.4.3 Shear and Torsion Design (on page 1497) for additional information.  
**Note:** This parameter is not used with ACI 318-14. |
| SKY            | 1.0           | Effective length factor for moment magnification procedure calculations for columns, in the local Y direction.  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| SKZ            | 1.0           | Effective length factor for moment magnification procedure calculations for columns, in the local Z direction.  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| SLY            | n/a           | Number of the load case that applies sidesway load in the local Y axis for column design.  
The moments from this case are used to magnify the local Y axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the SWY parameter setting.  
**Note:** This parameter is not used if MMAG has been specified.  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| SLZ            | n/a           | Number of the load case that applies sidesway load in the local Z axis for column design.  
The moments from this case are used to magnify the local Z axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the SWY parameter setting.  
**Note:** This parameter is not used if MMAG has been specified.  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| SQY            | 0.2           | Stability index, Q, in the local Y axis as defined in ACI 318 section 6.6.4.3(b).  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| SQZ            | 0.2           | Stability index, Q, in the local Z axis as defined in ACI 318 section 6.6.4.3(b).  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
| SWY            | 0             | Braced against sidesway:  
0) Not braced  
1) Braced  
**Note:** Use only when MMAG parameter is equal 0.  
**Note:** Only used with ACI 318-11 and ACI 318-14. |
## Parameter Name | Default Value | Description
--- | --- | ---
TRACK | 0.0 | Output detail
- Beam Design:
  0) Critical moment will not be printed out with beam design report.
  1) Critical moment will be printed out with beam design report
  2) Print out required steel areas for all intermediate sections specified by NSECTION.
- Column Design:
  0) Prints out detailed design reports
  1) Prints out column interaction analysis results in addition to TRACK 0.0 output.
  2) Prints out a schematic interaction diagram and intermediate interaction values in addition to TRACK 2.0 results.

TRN | 0 | Transverse loads between supports
  0 - False
  1 - True

Note: Only used with ACI 318-14.

WIDTH | ZD* | Width of the member measured in the local Z axis.

Note: This parameter is not used with ACI 318-14.

### Notes
- These values must be provided in the current unit system being used.

Notes

a. The value used when specifying the CLS parameter for column design is taken to be the clear cover for the longitudinal bars in a column. It is not taken as the clear cover for the tie bars. Therefore, the distance from the edge of the column to the centerline of the first row of longitudinal bars is CLS plus half the diameter of the main bar.

b. When using metric units for ACI design, provide values for these parameters in actual 'mm' units instead of the bar number. The following metric bar sizes are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm, and 60 mm.

c. Clear cover values are set for the concrete Not exposed to weather or in contact with ground. Plates or slabs are assumed to be reinforced with bar number 11 (in imperial units) or smaller.

d. Required for bar detailing in a physical member which might contain intermediate members along with cantilevers.

For internal members both STRS and EDSP should be set to 0.
e. In ACI 318-14, members are always designed with 5 sections. The parameter NSECTION used with older editions of the code has no effect.

**D1.F.3.2 Pre ACI 318-2011 Design Parameters**

The following outlines the parameters that are available to control the design of members as beams or columns per the ACI 318-99 through ACI 318-2008 codes. The commands that initiate the design of these is outlined in TR53.3 Concrete Design Command (on page 2860). If any parameter is not specified and its value is required in the design, then it will use that specified as the default value in the tables below.

**Note:** Not all the parameters are used in all versions of the ACI 318 code. Ensure that all the necessary parameters for the version of the code are properly specified. Where practical, parameters used in earlier versions of the code will be supported in the newer versions. Some old parameters are not supported in newer versions of the code and new parameters have been added in newer versions of the code that provide greater flexibility in the design.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CLB</td>
<td>1.5 in. (40 mm) for beams* 0.75 in. (20 mm) for plate elements*</td>
<td>Minimum cover from the base of the section to the bottom longitudinal reinforcement. See Note c on page 1495 below.</td>
</tr>
<tr>
<td>CLS</td>
<td>1.5 in. (40 mm)*</td>
<td>Minimum cover from the sides of the section to the side longitudinal reinforcement. See Note a on page 1486 and Note c on page 1495 below.</td>
</tr>
<tr>
<td>CLT</td>
<td>1.5 in. (40 mm) for beams* 0.75 in. (20 mm) for plate elements*</td>
<td>Minimum cover from the top of the section to the top longitudinal reinforcement. See Note c on page 1495 below.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD*</td>
<td>Depth of the member measured in the local Y axis. Note: This parameter is not used with ACI 318-14.</td>
</tr>
<tr>
<td>EFACE</td>
<td>0.0*</td>
<td>Face of support location at end of beam. If specified, the shear force at end is computed at a distance of EFACE + d from the end joint of the member. Note: See D1.F.4.3 Shear and Torsion Design (on page 1497) for additional information. Note: This parameter is not used with ACI 318-14.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>FC</td>
<td>4,000 psi (28 N/mm²)*</td>
<td>Compressive strength of concrete.</td>
</tr>
<tr>
<td>FYMAIN</td>
<td>60,000 psi (420 N/mm²)*</td>
<td>Yield stress for main longitudinal reinforcing steel.</td>
</tr>
<tr>
<td>FYSEC</td>
<td>60,000 psi (420 N/mm²)*</td>
<td>Yield stress for transverse reinforcing steel.</td>
</tr>
</tbody>
</table>
| LWF            | 1.0           | Modification factor, $\lambda$, for lightweight concrete as specified in ACI i318-08, cl. 8.6.1. Valid entries are between 0.75 and 1.0, inclusive. Used as a reduction factor for the mechanical properties of lightweight concrete.  
**Note:** This parameter is used in ACI 318-2008, 2001, and 2014 only. |
| MAXMAIN        | #18 bar (57 mm) | Maximum size of reinforcing bar to be used for longitudinal reinforcement.  
**Note:** See [Note b](on page 1495) below. |
| MINMAIN        | #4 bar (13 mm) | Minimum size of reinforcing bar to be used for longitudinal reinforcement.  
**Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below.  
**Note:** See [Note b](on page 1495) below. |
| MINSEC         | #4 bar (13 mm) | Minimum size of reinforcing bar to be used for transverse reinforcement.  
**Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below.  
**Note:** See [Note b](on page 1495) below. |
| MMAG           | 1.0           | Factor to magnify the analysis moments used for column design.  
**Note:** For ACI 318-14, if this is set to 0, then the moments for the local Y and Z axis will be calculated independently using the parameters BDY and BDZ. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NSECTION</td>
<td>12</td>
<td>Number of equally spaced sections to be considered in finding critical moments for beam design. The valid range is from 12 to 20. This parameter applies to the design process rather than individual members and thus should not have any assigned member list. If more than one NSECTION parameter is defined, then highest value will be used.</td>
</tr>
<tr>
<td>REINF</td>
<td>0</td>
<td>Column transverse reinforcement type. 0) links (tied column) 1) spiral reinforcement</td>
</tr>
<tr>
<td>RHOMIN</td>
<td>0.01</td>
<td>Minimum reinforcement required in a concrete column. Enter a value between 0.0 and 0.08, where 0.08 = 8% reinforcement; the maximum allowed by the ACI code.</td>
</tr>
<tr>
<td>SFACE</td>
<td>0.0*</td>
<td>Face of support location at start of beam. If specified, the shear force at start is computed at a distance of SFACE + d from the start joint of the member.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Output detail • Beam Design: 0) Critical moment will not be printed out with beam design report. 1) Critical moment will be printed out with beam design report 2) Print out required steel areas for all intermediate sections specified by NSECTION • Column Design: 0) Prints out detailed design reports 1) Prints out column interaction analysis results in addition to TRACK 0.0 output. 2) Prints out a schematic interaction diagram and intermediate interaction values in addition to TRACK 2.0 results.</td>
</tr>
</tbody>
</table>

Note: This parameter is not used with ACI 318-14.
### Parameter Name | Default Value | Description
--- | --- | ---
WIDTH | ZD* | Width of the member measured in the local Z axis.

**Note:** This parameter is not used with ACI 318-14.

**Notes**

- **a.** The value used when specifying the CLS parameter for column design is taken to be the clear cover for the longitudinal bars in a column. It is not taken as the clear cover for the tie bars. Therefore, the distance from the edge of the column to the centerline of the first row of longitudinal bars is CLS plus half the diameter of the main bar.
- **b.** When using metric units for ACI design, provide values for these parameters in actual 'mm' units instead of the bar number. The following metric bar sizes are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm, and 60 mm.
- **c.** Clear cover values are set for the concrete not exposed to weather or in contact with ground. Plates or slabs are assumed to be reinforced with bar number 11 (in imperial units) or smaller.
- **d.** Required for bar detailing in a physical member which might contain intermediate members along with cantilevers.

For internal members both STRS and EDSP should be set to 0.

#### D1.F.4 Beam Design

Beams are designed at a series of locations along their length for the moment about the local Z axis (MZ), shear force in the local Y and Z axes (FY and FZ), torsional moment (MX), and axial force (FX) for an envelope of the forces from a specified collection of load cases and combinations.

The envelope is created at each design section from a collection of the max/min forces in all degrees of freedom for all the load cases and/or combinations that have been included in the design. As there are 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) and to capture the maximum and minimum effects, a total of 12 sets of loads are designed for. Each set will include the maximum or minimum of one force such as Max FX along with all the other associated forces in the load case/combination.

For each member that is designed as a beam, the forces from the current LOAD LIST (see TR.39 Load List Specification on page 2836)) will be used to create an envelope of forces.

This envelope of forces is then used to determine a required area of longitudinal rebar at each of the key section locations. Five sections are defined on each member: the start, quarter point, mid-point, three-quarter point, and end point. This required area is then used determine an optimized bar arrangement that meets the code requirements, including detailing rules. Once the longitudinal bars have been designed and detailed, the program then designs the transverse steel based on the code provisions. Note that in case of torsion loads, the effect of the applied torsion will be catered for while designing the longitudinal steel as well as the transverse steel.

The required area of reinforcements at every cross section is first calculated. In order to then arrive at an optimized detailed bar layout, the program starts with the bar size defined in the MINMAIN parameter (or MIMT parameter for top bars and MIMB parameter for bottom bars, if specified)

The program will then iterate though increasing the number of bars or the size of bars dependent on the values of the BRDO and TRDO parameters.

If additional skin bars (i.e., side face bars) are needed for spacing requirements, these will be added.
Any detailing rules such as development lengths etc will be considered and the final design output would indicate whether a given set of bars are not be anchored or not.

**Assumed Values Used**

Most of the values are determined from the beam data, however, some values of note include:

- Young’s modulus of rebar \((E_s)\) is taken as 29,000 ksi
- Young’s modulus of concrete is calculated from the given value of \(F'_c\). But if \(F'_c > 10,000\), then \(E_c\) is limited to 3,605 psi.
- Density of concrete is taken as 145 pcf (0.083912 kip-in\(^3\))

**Scope**

Note that deep beams – where the span < 4× the beam depth – are not designed. A warning is reported in the output.

**Related Links**

- *V. ACI 318-14 Tee Beam* (on page 4272)

**D1.F.4.1 Strength and Ductility Design**

The strength design is performed in two steps considering the longitudinal reinforcement design for axial-bending in an initial step, and the shear and torsion design including the longitudinal reinforcement for torsion tension in a latter pass together with the transverse design.

The member resistance is calculated based on strain compatibility and interaction surface calculations.

The following conditions should be checked (9.5.1.1)

\[
\phi M_n \geq M_u \\
\phi V_n \geq V_u \\
\phi T_n \geq T_u \\
\phi P_n \geq P_u
\]

\(\phi\) is determined according to 21.2

The ratio of neutral axis depth to the depth of the furthest rebar in tension is limited such that the section strain at the location of the maximum rebar depth is a minimum of 0.004.

Application of the ductility checks are limited to cross sections with net axial load (compression) less than 0.10\(f'_c\) Ag, in accordance with section 9.3.3.1.

The maximum strain in the tension reinforcement is calculated and compared with the Code limit. In order to reduce the time involved in the axial-bending calculations, the approximation may currently have a maximum difference with the more precise values of 12% mostly in biaxial cases, the approximation will be much better for other cases.

**D1.F.4.2 Minimum Flexural Reinforcement**

The code specifies to dispose a minimum area of flexural reinforcement where tension reinforcement is required.

As a minimum bound, the condition that is given in 9.6.2.1 is used. Although it is intended only for prestressed beams, it may be used also in reinforced concrete beams and control in odd circumstances such as where the
specified cover is extremely large. The bending strength of the section is designed to be at least \(1.2 \, M_{cr}\), where the cracking moment is assumed to be applied to each axis separately as uniaxial bending. No simultaneous biaxial moments are considered. The cracking moments about each axis are calculated to cause the stress in the extreme tension fiber to reach \(f_r\). In order to account for a possible varying of elastic modulus \((E_c)\) values, the following formula is used ignoring the effect of the inertia product \((I_{xy})\):

\[
M_{cr(region)} = \frac{\sigma_{cr(region)} E_c I_c}{E_c f_r(region)}
\]

where

- \(\sigma_{cr(region)}\) = the flexural tension strength for a region
- \(E_c(region)\) = the modulus of elasticity for a specific region
- \(c(region)\) = the distance from the neutral axis to the extreme fiber of a region
- \(E_c I_c\) = the integrated product of the inertia and elastic modulus for the entire section
- \(f_r\) = the modulus of rupture calculated in accordance with Clause 19.2.3.1
- \(\lambda\) = calculated according to Clause 19.2.4.2

The cracking moment is individually calculated for each axis and only considers a particular axis if there is a considerable bending moment about that axis. The sign of the acting moment is also considered.

This minimum bound is considered only when the 9.6.1.2 or 9.6.1.3 is also applied.

D1.F.4.3 Shear and Torsion Design

In the absence of torsion, the required shear bar density at cross section at each cross section will initially be calculated for shear along both the local Z and Y axes. Based on these required shear densities, the program will divide the member into one or more shear zones such that each zone (between the start and end cross sections) will require the same shear density. Once the shear zones have been identified, transverse bar links will be provided based on the bar size specified using the MINSEC parameter. In order to meet the required density demand, the program will start with the minimum number of legs and calculate the most optimum spacing for the transverse links. If the minimum / maximum spacing criteria cannot be met during this process, additional legs will be added and the spacings adjusted to optimize the design. Note that the program will use the same bar size for links to cater for shear along both the X and Z directions to maintain a practical design.

Note that the program will use the same bar size for all transverse bars to cater for shear requirements in both axes to maintain a practical design.

If the beam is also subject to torsional forces \((MX)\), both the longitudinal steel as well the transverse steel will be designed to cater for the applied torsion, in addition to any longitudinal or transverse steel that would have been required for the bending/shear effects. The torsion force increases the demand in the transverse due to pure shear and longitudinal direction due to warping effects.

See section F.7 for the clauses considered for shear and torsion design.

D1.F.4.4 Definition of Bar Positions

Both input parameters and design output refer to the positions of bars in beam sections as follows.

**Note:** The bars shown in these figures are representational only; they do not correspond to any examples in this section.
D1.F.4.5 Beam Design Output

The following report options are provided for the output for a beam design.

TRACK 0 Output

Setting the option TRACK 0 will cause the program to produce the required area of longitudinal as well as transverse steel at 5 equally spaced locations –including member ends– along the member length.

**Note:** This option is meant to be considered for quick area of steel (As) requirement analysis and should *not* be considered as a safe, acceptable design.

Sample TRACK 0 output:

**Member** member number
Type: Beam
Shape: Either rectangular or Tee
As: the area of longitudinal steel required at each cross section
As/sv: the transverse steel density required at each cross section

STAAD.PRO CONCRETE DESIGN - (ACI-318-14) v2.0
****************************************************
Units: Kip and Inch (Unless Noted Otherwise)

| Member : 12 | Type : Beam | Shape : Rectangular |
|----------------------------------------------------------|
| AREA OF STEEL REQUIRED | | |
|----------------------------------------------------------|
| Section - | 1 | 2 | 3 | 4 | 5 |
| Location | 0.00 | 60.00 | 120.00 | 180.00 | 240.00 |
| As(Longitudinal) | 7.17 | 4.46 | 4.94 | 4.46 | 3.95 |
| As/sv(Trans Y) | 0.09 | 0.05 | 0.01 | 0.02 | 0.07 |
| As/sv(Trans Z) | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

TRACK 1 Output

Setting the program to output the beam design with a TRACK 1 option will perform full design and detailing operations and will report a summary of the design. A TRACK 1 report will include the following sections:

Design Summary
This section provides a summary of the design including the design status (i.e., PASS/FAIL), member type, length, the critical ratio, the critical criteria, and relevant clauses.

Cross section details
This section provides shape type and dimensions of the cross section profile.

Longitudinal Bar Layout
The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of "yes" in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed fit. A value of "no" indicates that no additional extension to the bar is needed beyond the given dimension. The value given in "Distance from face" is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.

Transverse Bar Layout
The transverse bar table reports the details of various shear zones (see D1.F.4.3 Shear and Torsion Design (on page 1497) ) within the span for both the local Y & Z axes.

Each zone is identified by the unique start and end location reported as the distances “From” and “To.”

This provides shear densities as minimum required and as provided by the given bar details.

The value "Nuns" is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.
Note: This output will not report any warnings or errors from the design and member analysis. These are only reported in the TRACK 2 output.

Sample beam design TRACK 1 output:

STAAD.PRO CONCRETE DESIGN - (ACI-318-14) v2.0
***********************************************
Units: Kip and Inch (Unless Noted Otherwise)

Member : 13

DESIGN SUMMARY

<table>
<thead>
<tr>
<th>Status</th>
<th>Fail</th>
<th>Type : Beam</th>
<th>Length: 216.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>1.000</td>
<td>Criteria: Torsion</td>
<td></td>
</tr>
<tr>
<td>Critical Clause</td>
<td>9.5.3/9.5.4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>All Failures</td>
<td>N.A</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

CROSS SECTION

<table>
<thead>
<tr>
<th>Shape: Rectangular</th>
<th>Width: 16.00</th>
<th>Depth: 21.00</th>
</tr>
</thead>
</table>

LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars Nums</th>
<th>Size</th>
<th>Location Start</th>
<th>Location End</th>
<th>Distance From Face</th>
<th>Anchor Start End</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right</td>
<td>2 #11</td>
<td></td>
<td>0.00</td>
<td>216.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Top</td>
<td>2 #11</td>
<td></td>
<td>0.00</td>
<td>216.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Top</td>
<td>1 #11</td>
<td></td>
<td>121.73</td>
<td>216.00</td>
<td>2.20</td>
<td>No Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>2 #11</td>
<td></td>
<td>0.00</td>
<td>216.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Left</td>
<td>2 #11</td>
<td></td>
<td>0.00</td>
<td>216.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
</tbody>
</table>

TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir</th>
<th>From</th>
<th>To</th>
<th>Asv Density Reqd.</th>
<th>Prov.</th>
<th>Rebar Specification Nums</th>
<th>Size</th>
<th>Spacing</th>
<th>Legs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>54.00</td>
<td>0.06</td>
<td>0.07</td>
<td>10 # 4</td>
<td>6.00</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>54.00</td>
<td>0.02</td>
<td>0.05</td>
<td>8 # 4</td>
<td>7.71</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Y</td>
<td>54.00</td>
<td>108.00</td>
<td>0.03</td>
<td>0.05</td>
<td>8 # 4</td>
<td>7.71</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Z</td>
<td>54.00</td>
<td>108.00</td>
<td>0.02</td>
<td>0.05</td>
<td>8 # 4</td>
<td>7.71</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Y</td>
<td>108.00</td>
<td>216.00</td>
<td>0.10</td>
<td>0.12</td>
<td>34 # 4</td>
<td>3.27</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Z</td>
<td>108.00</td>
<td>216.00</td>
<td>0.02</td>
<td>0.09</td>
<td>26 # 4</td>
<td>4.32</td>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

TRACK 2 Output

TRACK 2 will produce the results of TRACK 1 along with the details of the bars and their distribution at each cross section. This option will also report any of the relevant detailed design messages. The longitudinal bar
details at each cross section will have information on the setting out of each bar at the specific cross section. This should serve as a useful tool to inspect the design in detail. A TRACK 2 output would contain:

**Design Summary**
This section provides a summary of the design including the design status, i.e. PASS/FAIL, member type and length, the critical ratio, the critical criteria and relevant clauses.

**Cross Section Details**
This section provides the shape type and dimension(s) of the cross-section.

**Design Inputs**
This section reports the material data for the steel and concrete including the yield strength for steel (FY), compressive strength of concrete (FC) and modulus of elasticity for both (Es and Ec) This section additionally includes details of the cover to the longitudinal bars from the top, bottom and side faces.

**Critical Strength Results**
This section reports the demand determined from the design envelope as described in DLF 4 Beam Design (on page 1495), along with the capacity for forces applied in positive and negative directions. The ratio is then the value of the demand divided by the appropriate capacity.

**Longitudinal Bar Details at Cross Sections**
For each of the design cross sections, the longitudinal bars are reported for each face where they are required, top bottom and skin. Area of steel is reported as the optimum required from the design and that provided after additionally meeting the detailing requirements.

The total quantity of bars on the given face are reported as “No(s) bars,” the size of longitudinal bars and quantity of layers that the bars are to be distributed on. For example, if the number of bars reported is 10 and in 2 layers, then there will be 5 bars in each layer.

**Longitudinal Bar Layout**
The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of “yes” in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed fit. A value of “no” indicates that no additional extension to the bar is needed beyond the given dimension. The value given in “Distance from face” is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.

**Transverse Bar Layout**
The transverse bar table reports the details of various shear zones (see DLF 4.3 Shear and Torsion Design (on page 1497)) within the span for both the local Y & Z axes.

Each zone is identified by the unique start and end location reported as the distances “From” and “To.”

This provides shear densities as minimum required and as provided by the given bar details.

The value “Nums” is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.

**Note:** This output will additionally report any warnings or errors from the design and member analysis.

Sample beam design TRACK 2 output:

```
STAAAD.PRO CONCRETE DESIGN - (ACI-318-14) v2.0
**************************************************************************
Units: Kip and Inch (Unless Noted Otherwise)
```
Member: 14

DESIGN SUMMARY

<table>
<thead>
<tr>
<th>Status</th>
<th>Pass</th>
<th>Type</th>
<th>Beam</th>
<th>Length: 240.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>0.832</td>
<td>Criteria:</td>
<td>Shear Y</td>
<td></td>
</tr>
<tr>
<td>Critical Clause</td>
<td>9.5.3/9.5.4</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

CROSS SECTION

<table>
<thead>
<tr>
<th>Shape</th>
<th>Rectangular</th>
<th>Width: 16.00</th>
<th>Depth: 21.00</th>
</tr>
</thead>
</table>

DESIGN INPUTS

| Concrete | Fc     | 4.000 |  |
| Steel    | Fy(main) | 60.000 | Fy(trans) | 60.000 |  |
| Cover    | Top     | 1.500 | Bottom    | 1.500 | Sides: 1.500 |

CRITICAL STRENGTH RESULTS

<table>
<thead>
<tr>
<th>Category</th>
<th>Demand</th>
<th>Min Capacity</th>
<th>Max Capacity</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>9.117</td>
<td>-961.359</td>
<td>673.920</td>
<td>0.014</td>
</tr>
<tr>
<td>Flexure</td>
<td>-4462.279</td>
<td>-5808.514</td>
<td>5086.883</td>
<td>0.768</td>
</tr>
<tr>
<td>Shear Y</td>
<td>97.255</td>
<td>-116.842</td>
<td>116.842</td>
<td>0.832</td>
</tr>
<tr>
<td>Shear Z</td>
<td>-0.636</td>
<td>-22.857</td>
<td>22.857</td>
<td>0.028</td>
</tr>
<tr>
<td>Torsion</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

LONGITUDINAL BAR DETAILS AT CROSS SECTIONS

<table>
<thead>
<tr>
<th>Distance</th>
<th>Position</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>Top</td>
<td>4.572</td>
<td>4.680</td>
<td>3</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>Bottom</td>
<td>0.672</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Left</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Right</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>60.000</td>
<td>Top</td>
<td>1.524</td>
<td>4.680</td>
<td>3</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>Bottom</td>
<td>1.002</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Left</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Right</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>120.000</td>
<td>Top</td>
<td>1.524</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>Bottom</td>
<td>1.484</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Left</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Right</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>180.000</td>
<td>Top</td>
<td>1.524</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>Bottom</td>
<td>1.002</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Left</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Right</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>240.000</td>
<td>Top</td>
<td>1.524</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>Bottom</td>
<td>0.495</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Left</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Right</td>
<td>0.966</td>
<td>3.120</td>
<td>2</td>
<td>#11</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars Nums</th>
<th>Size #11</th>
<th>Location Start</th>
<th>Location End</th>
<th>Distance From Face</th>
<th>Anchor Start End</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right</td>
<td>2</td>
<td></td>
<td>0.00</td>
<td>240.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Top</td>
<td>2</td>
<td>#11</td>
<td>0.00</td>
<td>240.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Top</td>
<td>1</td>
<td>#11</td>
<td>0.00</td>
<td>108.54</td>
<td>2.20</td>
<td>Yes No</td>
</tr>
<tr>
<td>Bottom</td>
<td>2</td>
<td>#11</td>
<td>0.00</td>
<td>240.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Left</td>
<td>2</td>
<td>#11</td>
<td>0.00</td>
<td>240.00</td>
<td>2.20</td>
<td>Yes Yes</td>
</tr>
</tbody>
</table>

TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir.</th>
<th>From</th>
<th>To</th>
<th>Asv Density</th>
<th>Rebar Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Req'd</td>
<td>Prov'd</td>
</tr>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>120.00</td>
<td>0.12</td>
<td>0.14</td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>120.00</td>
<td>0.03</td>
<td>0.10</td>
</tr>
<tr>
<td>2</td>
<td>Y</td>
<td>120.00</td>
<td>240.00</td>
<td>0.10</td>
<td>0.10</td>
</tr>
<tr>
<td>2</td>
<td>Z</td>
<td>120.00</td>
<td>240.00</td>
<td>0.03</td>
<td>0.10</td>
</tr>
</tbody>
</table>

-D1.F.4.6 Beam Design per ACI 318-11 and Earlier-

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to locate the possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with an NSECTION parameter. All of these equally spaced sections are scanned to determine moment and shear envelopes.

-D1.F.4.6.1 Cracked Moment of Inertia - ACI Beam Design-

When beam design is done per ACI 318, STAAD will report the moment of inertia of the cracked section at the location where the design is performed. The cracked section properties are calculated in accordance with the equations shown below.

Rectangular Sections
Figure 161: Gross section (A) and cracked transform section (B) for rectangular shapes

Without compression steel

\[ n = \frac{E_I}{E_c} \]

\[ B = \frac{b}{nA_I} \]

\[ I_g = \frac{b \times h^3}{12} \]

\[ kd = \frac{\sqrt{2kd \times B + 1}}{B} - 1 \]

\[ I_{cr} = \frac{b(bkd)^3}{3} + nA_s(d - kd)^2 \]

Tee Shaped Sections
Without compression steel

\[ C = \frac{b_w}{nA_s} \]

\[ f = \frac{h_f(b - b_w)}{nA_s} \]

\[ y = h - \frac{1}{2} \left( \frac{b - b_w}{b_w} \right) h_f^2 + b_w h_f + b_w h \]

\[ kd = \frac{C(2d + h_f f) + (1 + f)^2 - (1 + f)}{C} \]

\[ I_{cr} = \frac{(b - b_w) h_f^3}{12} + \frac{b_w (kd)^3}{3} + \left( b - b_w \right) h_f \left( kd - \frac{h_f}{2} \right)^2 + nA_s (d - kd)^2 \]

See D1.F.4.5 Beam Design Output (on page 1498) for an example of output including the calculated cracked moment of inertia.

D1.F.4.6.2 Design of I-shaped beams per ACI-318

I-shaped sections can be designed as beams per the ACI 318 code. The property for these sections must be defined through a user table, I-section, or using the tapered specification. Information on assigning properties in this manner is available in TR.19 User Steel Table Specification (on page 2446) (I-section type) and TR.20.3 Tapered Member Specification (on page 2468) (Tapered I shape) of the Technical Reference manual.

From the standpoint of the analysis – determining member forces, nodal displacements and support reactions – the same set of facilities and rules which are applicable for any normal reinforced concrete frames or other structures can be used when I-sections or tapered concrete members are specified. In other words, there isn’t anything unique or special to account for in the analysis model simply because I-shaped concrete beams are part of it.

From the standpoint of design, the following rules are applicable:

---

Figure 162: Gross and cracked transform sections for tee shapes without compression steel
1. The member can be designed as a beam using the general principles explained in Chapter 3 of the Technical Reference manual. It currently cannot be designed as a column. Design as a beam is done for flexure (MZ), shear (FY) and torsion (MX) just like that for rectangular, tee or trapezoidal beams. Axial forces (FX) are used during the capacity computations in shear and torsion. At each section along the length that the member is designed at, the depth at that section location is used for effective depth computation.

2. The program performs the following tests on the section dimensions before starting the design:

- If the thickness of the web is the same as the width of the top and bottom flanges, the member is designed as a rectangular section.
- If the thickness of the web is the same as the width of one of the flanges but not the other, the member is designed as a T-section or a rectangular section, depending on which side the compression due to bending is at.
- If the web thickness does not match the width of either flange, design is done using the rules applicable for T-beams – one flange is in compression, the other in tension, and tensile capacity of concrete on the tensile side of the neutral axis is ignored.
- The program is also able to design the beam as a doubly reinforced section if it is unable to design it as a single-reinforced section.

3. The parameters for designing these members are as shown in DLF.3 Design Parameters (on page 1481) of this manual. Detailed output on design at individual section locations along the member length may be obtained by setting the TRACK parameter to 3.0.

An example for I-beam design is shown below.

```
STAAD PLANE I BEAM CONCRETE DESIGN PER ACI-318
UNIT FEE KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0
MEMBER INCIDENCES
1 1 2
UNIT INCHES KIP
MEMBER PROPERTY
1 TAPERED 18 10 18 15 2.5
CONSTANTS
E 3300 ALL
DENSITY CONCRETE ALL
POISSON CONCRETE ALL
SUPPORTS
1 2 PINNED
UNIT FEET KIP
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI GY -5.76
LOAD 2 LIVE LOAD
1 UNI GY -7.04
LOAD COMB 3 ACI 318-02
1 1.4 2 1.7
PERFORM ANALYSIS
LOAD LIST 3
START CONCRETE DESIGN
CODE ACI 2002
UNIT INCHES KIP
MINMAIN 9 ALL
FC 4 ALL
FYMAIN 60 ALL
TRACK 2.0 ALL
```
D1.F.4.6.3 ACI 318-11 and Earlier Beam Design Output

The following annotations apply to the output:

**LEVEL** Serial number of bar level which may contain one or more bar group
**HEIGHT** Height of bar level from the bottom of beam
**BAR INFO** Reinforcement bar information specifying number of bars and bar size
**FROM** Distance from the start of the beam to the start of the reinforcement bar
**TO** Distance from the start of the beam to the end of the reinforcement bar
**ANCHOR(STA/END)** States whether anchorage, either a hook or continuation, is needed at start (STA) or at the end

**ROW** Actually required flexural reinforcement ($A_s/bd$) where $b =$ width of cross section ($ZD$ for rectangular and square section) and $d =$ effective depth of cross section ($YD$ - distance from extreme tension fiber to the c.g. of main reinforcement).

**ROWMN** Minimum required flexural reinforcement ($A_{min}/bd$)
**ROWMX** Maximum allowable flexural reinforcement ($A_{max}/bd$)
**SPACING** Distance between centers of adjacent bars of main reinforcement
**Vu** Factored shear force at section
**Vc** Nominal shear strength provided by concrete
**Vs** Nominal shear strength provided by shear reinforcement
**Tu** Factored torsional moment at section
**Tc** Nominal torsional moment strength provided by concrete
**Ts** Nominal torsional moment strength provided by torsion reinforcement

---

The following is a sample TRACK 1.0 output from concrete beam design per ACI (from the file C:sers\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US.8. Concrete Design for a Space Frame.std):

---
**Beam No. 14 Design Results - Flexure Per Code ACI 318-05**

- **Length (LEN):** 20.00ft.
- **Fy:** 60000.
- **Ft:** 4000.
- **Size:** 16.00 X 21.00 inches

<table>
<thead>
<tr>
<th>Level</th>
<th>Height</th>
<th>Bar Info</th>
<th>From</th>
<th>To</th>
<th>Anchor</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FT.</td>
<td>IN.</td>
<td>FT.</td>
<td>IN.</td>
<td>STA</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>+2-3/8</td>
<td>8-NUM.5</td>
<td>2  +2-5/8</td>
<td>20 + 0-0/0</td>
</tr>
<tr>
<td></td>
<td>CRITICAL POS MOMENT=</td>
<td>191.70 KIP-FT</td>
<td>AT 11.67 FT</td>
<td>LOAD 1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>REQD STEEL=</td>
<td>2.47 IN2, RHO=0.0083, RHOMX=0.0214 RHOMN=0.0033</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>MAX/MIN/ACTUAL BAR SPACING=</td>
<td>10.00/ 1.62/ 1.62 INCH</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>REQD. DEVELOPMENT LENGTH =</td>
<td>27.37 INCH</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Cracked Moment of Inertia Iz at above location = 4320.43 inch^4

2
1 + 6-1/8 4-NUM.11 0 + 0-0/0 16 + 2-0/0  YES  NO

| CRITICAL NEG MOMENT= | 371.81 KIP-FT | AT 0.00 FT | LOAD 1 |
| REQD STEEL= | 5.39 IN2, RHO=0.0184, RHOMX=0.0214 RHOMN=0.0033 |
| MAX/MIN/ACTUAL BAR SPACING= | 10.00/ 2.82/ 3.53 INCH |
| REQD. DEVELOPMENT LENGTH = | 80.14 INCH |

Cracked Moment of Inertia Iz at above location = 8050.77 inch^4

3
1 + 6-3/8 3-NUM.6 16 +10-1/4 20 + 0-0/0  NO  YES

| CRITICAL NEG MOMENT= | 104.94 KIP-FT | AT 20.00 FT | LOAD 1 |
| REQD STEEL= | 1.30 IN2, RHO=0.0044, RHOMX=0.0214 RHOMN=0.0033 |
| MAX/MIN/ACTUAL BAR SPACING= | 10.00/ 1.75/ 5.62 INCH |
| REQD. DEVELOPMENT LENGTH = | 21.35 INCH |

Cracked Moment of Inertia Iz at above location = 2602.81 inch^4

**Beam No. 14 Design Results - Shear**

At Start Support - Vu= 84.31 KIP  Vc= 35.77 KIP  Vs= 76.64 KIP
Tu= 0.23 KIP-FT  Tc= 5.70 KIP-FT  Ts= 0.00 KIP-FT  LOAD 1
No Stirrups are required for torsion.
Reinforcement is required for shear.
Provide Num. 5 2-legged stirrups at 4.7 in. C/C for 102. in.

At End Support - Vu= 57.62 KIP  Vc= 35.77 KIP  Vs= 41.06 KIP
Tu= 0.23 KIP-FT  Tc= 5.70 KIP-FT  Ts= 0.00 KIP-FT  LOAD 1
No Stirrups are required for torsion.
Reinforcement is required for shear.
Provide Num. 4 2-legged stirrups at 9.3 in. C/C for 102. in.
D1.F.5 Column Design

Columns design as per ACI 318-14 is based on a full 3D interaction surface for a given cross section. Columns will be designed for the combined effects of axial loads and bi-axial bending moments. Other effects such as torsion and shear will also be accounted for in the design. The program then provides a final detailed design followed by a cross section wise analysis to ensure that the design is satisfactory. During the analysis stage the strengths of the cross section for each of these effects will also be calculated. The design process works on the basis of being able to find a point on or very close to the interaction surface so as to ensure the most efficient design.

As with beam design, an envelope is created at each design section from a collection of the max/min forces in all degrees of freedom for all the load cases and combinations that have been included in the design. As there are six degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) and to capture the maximum and minimum effects, a total of 12 sets of loads are designed for. Each set will include the maximum or minimum of one force such as Max FX along with all the other associated forces in the load case/combination.

The following section shapes will be considered for column design with ACI 318-14:

1. square
2. rectangular
3. circular
4. tee-shaped

All column designs will be based on a uniform distribution of reinforcement around the column perimeter. The design options to control a column design will be based on the CRDO parameter setting.

Related Links
- V. ACI 318-14 Circular Column (on page 4263)
- V. ACI 318-14 Rectangular Column (on page 4266)
- V. ACI 318-14 Square Column (on page 4269)

**D1.F.5.1 Strength Design**

All column designs will be based on a uniform distribution of reinforcement around the column perimeter. The design options to control a column design will be based on the CRDO parameter setting.

Column resistance is calculated based on strain compatibility and interaction surface calculations.

The following conditions should be checked (10.5.1.1)

\[ \phi P_n \geq P_u \]
\[ \phi M_n \geq M_u \]
\[ \phi V_n \geq V_u \]
\[ \phi T_n \geq T_u \]
\[ \phi P_n \geq P_u \]
\[ \phi M_n \geq M_u \]
\[ \phi V_n \geq V_u \]
\[ \phi T_n \geq T_u \]
Where the nominal axial compressive strength, \( P_n \), shall not exceed \( P_{n,\text{max}} \) in accordance with Table 22.4.2:

\[
P_{n,\text{max}} = 0.80 P_o \text{ for ties}
\]
\[
P_{n,\text{max}} = 0.85 P_o \text{ for spirals}
\]
\[
P_o = 0.85 f'c (A_g - A_{st}) + f_y A_{st})
\]

where

\( A_{st} \) = the total area (of non prestressed) longitudinal reinforcement
\( A_g \) = the gross area of the concrete section
\( f'c \) = the specified compressive strength of concrete
\( f_y \) = the specified yield strength of passive reinforcement
\( M_n \) = the nominal flexural strength
\( V_n \) = the nominal shear strength
\( T_n \) = the nominal torsional moment

The design is performed in two steps considering the longitudinal reinforcement design for axial-bending in an initial step, and the shear and torsion design including the longitudinal reinforcement for torsion tension in a latter pass together with the transverse design.

The minimum area of longitudinal reinforcement is taken as 0.01\( A_g \) and the maximum as 0.08\( A_g \). In agreement with the commentary a warning is triggered when the reinforcement ratio is over 4 percent due to possible lap splices zones.

**D1.F.5.2 Slenderness Effects and Analysis Considerations**

Slenderness effects are extremely important in designing compression members. The ACI 318-14 code specifies two options by which the slenderness effect can be accounted for. One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed-end moments, the effect of deflections on moments and forces, and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD.Pro uses both these options. To perform the first type of analysis, use the command \texttt{PDELTA ANALYSIS} instead of \texttt{PERFORM ANALYSIS}. This analysis method will accommodate the requirements as specified in Section 6.7 of the ACI 318-14 Code (Section 10.10 in previous editions), except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If it is desired, STAAD.Pro can also accommodate any arbitrary moment magnification factor (second option) as an input, in order to provide some safety due to the effects of the duration of loads.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, moment-magnification is based on empirical formula and assumptions on sidesway.

Considering all this information, it is our belief that a \texttt{PDELTA ANALYSIS}, as performed by STAAD.Pro, is most appropriate for the design of concrete members. However, you should note that to take advantage of this
analysis, all combinations of loadings must be provided as repeat load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a repeat load case consists of one or more primary load case which are revised during the PDelta analysis based on the deflections. Also note that you must provide the proper factored loads (e.g., 1.4 for DL etc.). STAAD.Pro does not factor the loads automatically.

**D1.F.5.3 Moment Magnification**

If the moment magnification method is used in lieu of an exact analysis, columns designed per ACI 318-14 in STAAD.Pro can consider sway using the moment magnification procedure. The second order effects are approximated by magnified moments for slender columns (as determined by clause 6.2.5/6.6.4.6) in sway and non-sway frames.

The following parameters apply to these calculations:

For sway frames, moments should be amplified twice, once for the member end sway (\(P\cdot\Delta\) effect) provided in clause 6.6.4.6 and using those amplified moments, again amplify them for member curvature along the length effects (\(P\cdot\delta\) effect) based on clause 6.6.4.5.

For non-sway frames moments need only be amplified once for the member curvature along the length effects (\(P\cdot\delta\) effect) based on clause 6.6.4.5

**STAAD Parameters Required to Perform the Design and Their Functionality**

- **SWY**  
  Is member braced against side-sway: This parameter checks whether the column being designed is braced structurally against side sway for lateral loads. It will affect the slenderness limit of the structure. This parameter is based on clause 6.2.5

- **BDY / BSZ**  
  \(\beta_{dns}\) factor required for Eq. 6.6.4.4.4(a) for stiffness reduction in case of non-sway case.

- **SQY / SQZ**  
  Stability index value which is required for two main purpose; one is for checking whether the moments need to be magnified for sway or non-sway case, Clause 6.6.4.3(b), and for the calculation of \(\delta_s\) give in Eq. 6.6.4.6.2(a) for moment amplification of a sway case.

- **SKY / SKZ**  
  Effective length factor which is based on Fig. R6.2.5 to be used to calculate member slenderness in Eq. 6.2.5(a)/(b)

- **SLY / SLZ**  
  The external load case number for the sway case which is used to extract moments \(M_{1s}\) and \(M_{2s}\) in equation 6.6.4.6.1(a)/(b). Incase this parameter is 0, it indicates to the program you do not want to include a load case which based on their judgement will not produce side-sway and hence needs to be amplified.

- **TRN**  
  This parameter asks from the user whether there are in transverse loads in the column between its two ends. It affects the calculation on Cm based on Eq. 6.6.4.5.3(a)/(b) which is required for amplification of moments for non-sway case.

- **MMAG**  
  This is the starting point parameter for the program. A non-zero value represents that the program will ignore all the moment amplification algorithm and just amplify the moments by multiplying this MMAG user-defined factor.

**Note:** The program will only go through with the moment magnification algorithm if and only if MMAG is specified as 0.

**Flowchart**

Given below is a flowchart of the moment amplification algorithm which is being incorporated in STAAD.Pro.
Design

D. Design Codes

- **MMAG = 0**: NO → **M_second = MMAG * M_first**

- **Do Not magnify moments**: NO → **SK * L / r > 22**
  - 6.2.5(a)
  - YES → **SWY = 1**
  - 6.4.3(b)
  - NO → **Do Not magnify moments**

- **NO** → **Q > 0.08**
  - 6.4.3(d)

- **YES** → **Magnify moments for ends which account for the P-Large Delta effect**
  - 6.6.4.6

- **NO** → **Magnify moments for P-Small Delta effects only**
  - 6.6.4.5

- Using the above magnified moments, we again magnify the end moments for member curvature along the length which accounts for the P-Small Delta effect.
  - 6.6.4.5

- **M_second <= 1.4 * M_first**: NO → **Throw STAAD warning**

- **YES** → **Do design with M_second**
D1.F.5.4 Definition of Bar Positions

Both input parameters and design output refer to the positions of bars in column sections as follows.

**Figure 164: ACI 318-14 Rectangular column bar positions**

Bars are evenly distributed around circumference

**Figure 165: ACI 318-14 Circular column bar positions**
D1.F.5.5 Column Design Output

The following report options are provided for the output for a column design.

TRACK 0 Output

Setting the option TRACK 0 will cause the program to produce the required area of longitudinal as well as transverse steel at 5 equally spaced locations—including member ends—along the member length.

**Note:** This option is meant to be considered for quick area of steel (As) requirement analysis and should not be considered as a safe, acceptable design.

Sample TRACK 0 output:

<table>
<thead>
<tr>
<th>Member</th>
<th>member number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Column</td>
</tr>
<tr>
<td>Shape</td>
<td>Either rectangular, Tee, or circular</td>
</tr>
<tr>
<td>As</td>
<td>the area of longitudinal steel required at each cross section</td>
</tr>
<tr>
<td>Asv/sv</td>
<td>the transverse steel density required at each cross section</td>
</tr>
</tbody>
</table>

### Sample TRACK 0 output:

<table>
<thead>
<tr>
<th>Section -</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>0.00</td>
<td>36.00</td>
<td>72.00</td>
<td>108.00</td>
<td>144.00</td>
</tr>
<tr>
<td>As(Longitudinal)</td>
<td>3.29</td>
<td>1.44</td>
<td>1.44</td>
<td>1.73</td>
<td>5.45</td>
</tr>
<tr>
<td>As/sv(Trans Y)</td>
<td>0.03</td>
<td>0.03</td>
<td>0.03</td>
<td>0.03</td>
<td>0.03</td>
</tr>
</tbody>
</table>

**Figure 166: ACI 318-14 Tee column bar positions**
TRACK 1 Output

Setting the program to output the beam design with a TRACK 1 option will perform full design and detailing operations and will report a summary of the design. A TRACK 1 report will include the following sections:

### Design Summary
This section provides a summary of the design including the design status (i.e., PASS/FAIL), member type, length, the critical ratio, the critical criteria, and relevant clauses.

### Cross section details
This section provides shape type and dimensions of the cross section profile.

### Longitudinal Bar Layout
The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of "yes" in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed fit. A value of "no" indicates that no additional extension to the bar is needed beyond the given dimension. The value given in "Distance from face" is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.

### Transverse Bar Layout
The transverse bar table reports the details of various shear zones (see D1.F.4.3 Shear and Torsion Design on page 1497) within the span for both the local Y & Z axes.

Each zone is identified by the unique start and end location reported as the distances "From" and "To."

This provides shear densities as minimum required and as provided by the given bar details.

The value "Nums" is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.

**Note:** This output will not report any warnings or errors from the design and member analysis. These are only reported in the TRACK 2 output.
LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Location</th>
<th>Distance</th>
<th>Anchor</th>
<th>Bars</th>
<th>Size</th>
<th>Start</th>
<th>End</th>
<th>Nums</th>
<th>#11</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>2.20</td>
<td>Yes</td>
<td>Top</td>
<td>0.00</td>
<td>144.00</td>
<td>3</td>
<td>11</td>
<td></td>
</tr>
<tr>
<td>Bottom</td>
<td>2.20</td>
<td>Yes</td>
<td>Bottom</td>
<td>0.00</td>
<td>144.00</td>
<td>3</td>
<td>11</td>
<td></td>
</tr>
<tr>
<td>Left</td>
<td>2.20</td>
<td>Yes</td>
<td>Left</td>
<td>0.00</td>
<td>144.00</td>
<td>1</td>
<td>11</td>
<td></td>
</tr>
<tr>
<td>Right</td>
<td>2.20</td>
<td>Yes</td>
<td>Right</td>
<td>0.00</td>
<td>144.00</td>
<td>1</td>
<td>11</td>
<td></td>
</tr>
</tbody>
</table>

TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir.</th>
<th>From</th>
<th>To</th>
<th>Reqd.</th>
<th>Prov.</th>
<th>Nums</th>
<th>Size</th>
<th>Spacing</th>
<th>Legs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>144.00</td>
<td>0.03</td>
<td>0.08</td>
<td>32</td>
<td>#4</td>
<td>4.65</td>
<td>2</td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>144.00</td>
<td>0.03</td>
<td>0.08</td>
<td>32</td>
<td>#4</td>
<td>4.65</td>
<td>2</td>
</tr>
</tbody>
</table>

TRACK 2 Output

TRACK 2 will produce the results of TRACK 1 along with the details of the bars and their distribution at each cross section. This option will also report any of the relevant detailed design messages. The longitudinal bar details at each cross section will have information on the setting out of each bar at the specific cross section. This should serve as a useful tool to inspect the design in detail. A TRACK 2 output would contain:

Design Summary
This section provides a summary of the design including the design status, i.e. PASS/FAIL, member type and length, the critical ratio, the critical criteria and relevant clauses.

Cross Section Details
This section provides the shape type and dimension(s) of the cross-section

Design Inputs
This section reports the material data for the steel and concrete including the yield strength for steel (FY), compressive strength of concrete (FC) and modulus of elasticity for both (Es and Ec). This section additionally includes details of the cover to the longitudinal bars from the top, bottom and side faces.

Critical Strength Results
This section reports the demand determined from the design envelope as described in D1.F.4 Beam Design (on page 1495), along with the capacity for forces applied in positive and negative directions. The ratio is then the value of the demand divided by the appropriate capacity.

Longitudinal Bar Details at Cross Sections
For each of the design cross sections, the longitudinal bars are reported for each face where they are required, top, bottom and skin. Area of steel is reported as the optimum required from the design and that provided after additionally meeting the detailing requirements.

The total quantity of bars on the given face are reported as “No(s) bars,” the size of longitudinal bars and quantity of layers that the bars are to be distributed on. For example, if the number of bars reported is 10 and in 2 layers, then there will be 5 bars in each layer.

Longitudinal Bar Layout
The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of “yes” in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed
A value of “no” indicates that no additional extension to the bar is needed beyond the given dimension. The value given in “Distance from face” is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.

The transverse bar table reports the details of various shear zones (see D1.F.4.3 Shear and Torsion Design (on page 1497)) within the span for both the local Y & Z axes.

Each zone is identified by the unique start and end location reported as the distances “From” and “To.”

This provides shear densities as minimum required and as provided by the given bar details.

The value “Nums” is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.

---

**STAAD.PRO CONCRETE DESIGN - (ACI-318-14) v2.0**

*Units: Kip and Inch (Unless Noted Otherwise)*

**Member : 10**

**DESIGN SUMMARY**

<table>
<thead>
<tr>
<th>Status</th>
<th>Pass</th>
<th>Type  : Column</th>
<th>Length: 144.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>0.942</td>
<td>Criteria: Flexure</td>
<td></td>
</tr>
<tr>
<td>Critical Clause:</td>
<td>10.5.2</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**CROSS SECTION**

<table>
<thead>
<tr>
<th>Shape: Rectangular</th>
<th>Width: 12.00</th>
<th>Depth: 12.00</th>
</tr>
</thead>
</table>

**DESIGN INPUTS**

<table>
<thead>
<tr>
<th>Concrete</th>
<th>Fc</th>
<th>4.000</th>
<th>Steel</th>
<th>Fy(main)</th>
<th>60.000</th>
<th>Fy(trans)</th>
<th>60.000</th>
<th>Ec</th>
<th>0.360E+04</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cover</td>
<td>Top</td>
<td>1.500</td>
<td>Bottom</td>
<td>1.500</td>
<td>Sides</td>
<td>1.500</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Design Messages**

---

**WARNINGS: DESIGN FOR MEMBER 10**

1) Ast Prov > 4 %, Cl – 10.6.1.1(commentary)
2) Incorrect shear min density or max spacing. Density may be ignored

**CRITICAL STRENGTH RESULTS**

<table>
<thead>
<tr>
<th>Category</th>
<th>Demand</th>
<th>Min Capacity</th>
<th>Max Capacity</th>
<th>Ratio</th>
</tr>
</thead>
</table>


LONGITUDINAL BAR DETAILS AT CROSS SECTIONS

<table>
<thead>
<tr>
<th>Distance</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>3.465</td>
<td>6.240</td>
<td>4</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>36.000</td>
<td>1.440</td>
<td>6.240</td>
<td>4</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>72.000</td>
<td>1.440</td>
<td>6.240</td>
<td>4</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>108.000</td>
<td>1.753</td>
<td>6.240</td>
<td>4</td>
<td>#11</td>
<td>1</td>
</tr>
<tr>
<td>144.000</td>
<td>5.650</td>
<td>6.240</td>
<td>4</td>
<td>#11</td>
<td>1</td>
</tr>
</tbody>
</table>

LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars Nums</th>
<th>Bars Size</th>
<th>Location Start</th>
<th>Location End</th>
<th>Distance From Face</th>
<th>Anchor Start End</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>2 #11</td>
<td>6.240</td>
<td>0.00</td>
<td>144.00</td>
<td>2.20</td>
<td>Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>2 #11</td>
<td>6.240</td>
<td>0.00</td>
<td>144.00</td>
<td>2.20</td>
<td>Yes</td>
</tr>
</tbody>
</table>

TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>144.00</td>
<td>0.03</td>
<td>0.10</td>
<td>38</td>
<td># 4</td>
<td>3.89</td>
<td>2</td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>144.00</td>
<td>0.03</td>
<td>0.10</td>
<td>38</td>
<td># 4</td>
<td>3.89</td>
<td>2</td>
</tr>
</tbody>
</table>

**D1.F.5.6 Column Design in Pre-2014 Codes**

Columns design in STAAD.Pro per the ACI code is performed for axial force and uniaxial as well as biaxial moments. All active loadings are checked to compute reinforcement. The loading which produces the largest amount of reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and circular sections, reinforcement is always assumed to be equally distributed on all faces. This means that the total number of bars for these sections will always be a multiple of four (4). If the MMAG parameter is specified, the column moments are multiplied by the MMAG value to arrive at the ultimate moments on the column. Since the ACI code no longer requires any minimum eccentricity conditions to be satisfied, such checks are not made.

Method used

Bresler Load Contour Method

Known Values

Pu, Muy, Muz, B, D, Clear cover, Fc, Fy

Ultimate Strain for concrete : 0.003
Steps involved

1. Assume some reinforcement. Minimum reinforcement (1%) is a good amount to start with.
2. Find an approximate arrangement of bars for the assumed reinforcement.
3. Calculate \( P_{\text{NMAX}} = 0.85 \, P_o \), where \( P_o \) is the maximum axial load capacity of the section. Ensure that the actual nominal load on the column does not exceed \( P_{\text{NMAX}} \). If \( P_{\text{NMAX}} \) is less than \( P_u/\Phi \), (\( \Phi \) is the strength reduction factor) increase the reinforcement and repeat steps 2 and 3. If the reinforcement exceeds 8%, the column cannot be designed with its current dimensions.
4. For the assumed reinforcement, bar arrangement and axial load, find the uniaxial moment capacities of the column for the Y and the Z axes, independently. These values are referred to as \( M_{\text{YCAP}} \) and \( M_{\text{ZCAP}} \) respectively.
5. Solve the interaction equation:

\[
\left( \frac{M_{ny}}{M_{\text{ycap}}} \right)^a + \left( \frac{M_{nz}}{M_{\text{zcap}}} \right)^a \leq 1.0
\]

Where \( a = 1.24 \)

If the column is subjected to a uniaxial moment, \( a \) is chosen as 1.0
6. If the Interaction equation is satisfied, find an arrangement with available bar sizes, find the uniaxial capacities and solve the interaction equation again. If the equation is satisfied now, the reinforcement details are written to the output file.
7. If the interaction equation is not satisfied, the assumed reinforcement is increased (ensuring that it is under 8%) and steps 2 to 6 are repeated.
8. The maximum spacing of reinforcement closest to the tension force, for purposes of crack control, is given by

\[
s = 15 \left( \frac{40,000}{f_s} \right) - 2.5 \leq 12 \left( \frac{40,000}{f_y} \right)
\]

with \( f_s \) in psi and is permitted to be taken equal to \((2/3) f_y\), rather than 60 percent of \( f_y \), as in ACI 318-02.
9. Section 10.9.3 has been modified to permit the use of spiral reinforcement with specified yield strength of up to 100,000 psi. For spirals with \( f_{yt} \) greater than 60,000 psi, only mechanical or welded splices may be used.

Column Interaction

The column interaction values may be obtained by using the design parameter \( \text{TRACK 1.0} \) or \( \text{TRACK 2.0} \) for the column member. If a value of 2.0 is used for the \( \text{TRACK} \) parameter, 12 different \( P_n-M_n \) pairs, each representing a different point on the \( P_n-M_n \) curve are printed. Each of these points represents one of the several \( P_n-M_n \) combinations that this column is capable of carrying about the given axis, for the actual reinforcement that the column has been designed for. In the case of circular columns, the values are for any of the radial axes. The values printed for the \( \text{TRACK 1.0} \) output are:

- \( P_0 \) = Maximum purely axial load carrying capacity of the column (zero moment).
- \( P_{\text{NMAX}} \) = Maximum allowable axial load on the column (Section 10.3.5 of ACI 318).
- \( P-\text{bal} \) = Axial load capacity at balanced strain condition.
- \( M-\text{bal} \) = Uniaxial moment capacity at balanced strain condition.
- \( e-\text{bal} \) = \( M-\text{bal} / P-\text{bal} \) = Eccentricity at balanced strain condition.
- \( M_0 \) = Moment capacity at zero axial load.
- \( P-\text{tens} \) = Maximum permissible tensile load on the column.
- \( \text{Des. } P_n \) = \( P_u/\Phi \) where \( \Phi \) is the Strength Reduction Factor and \( P_u \) is the axial load for the critical load case.
- \( \text{Des. } M_n \) = \( M_u*\text{MMAG}/\Phi \) where \( \Phi \) is the Strength Reduction Factor and \( M_u \) is the bending moment for the appropriate axis for the critical load case. For circular columns,
\[ M_u = \sqrt{M_{uy}^2 + M_{uz}^2} \]
\[ e/h = (M_u/P_n)/h \]

where
\[ h \quad \text{the length of the column} \]

Example
Column design per the ACI 318-2005 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2005
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example
Column design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example
Column design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Column Design Output

The samples illustrate different levels of the column design output. The following output is generated without any TRACK definition (i.e., using the default of TRACK 0.0):

```
====================================================================
COLUMN NO. 5 DESIGN PER ACI 318-05 - AXIAL + BENDING
FY - 60000 FC - 4000 PSI, SQRE SIZE - 12.00 X 12.00 INCHES, TIED
====================================================================
```
## Design

### D. Design Codes

### AREA OF STEEL REQUIRED

**AREA OF STEEL REQUIRED = 7.589 SQ. IN.**

### BAR CONFIGURATION

<table>
<thead>
<tr>
<th>BAR CONFIGURATION</th>
<th>REINF PCT.</th>
<th>LOAD</th>
<th>LOCATION</th>
<th>PHI</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 - NUMBER 9</td>
<td>5.556</td>
<td>2</td>
<td>STA</td>
<td>0.650</td>
</tr>
<tr>
<td>(PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TIE BAR NUMBER 4</td>
<td>SPACING 8.00 IN</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### TRACK 1.0

**TRACK 1.0 generates the following additional output:**

**COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)**

<table>
<thead>
<tr>
<th>P0</th>
<th>Pn max</th>
<th>P-bal.</th>
<th>M-bal.</th>
<th>e-bal. (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>942.40</td>
<td>753.92</td>
<td>179.59</td>
<td>170.75</td>
<td>11.41</td>
</tr>
<tr>
<td>M0</td>
<td>P-tens.</td>
<td>Des.Pn</td>
<td>Des.Mn</td>
<td>e/h</td>
</tr>
<tr>
<td>148.52</td>
<td>-480.00</td>
<td>350.15</td>
<td>10.47</td>
<td>0.00249</td>
</tr>
</tbody>
</table>

### COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)

<table>
<thead>
<tr>
<th>P0</th>
<th>Pn max</th>
<th>P-bal.</th>
<th>M-bal.</th>
<th>e-bal. (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>942.40</td>
<td>753.92</td>
<td>179.59</td>
<td>170.75</td>
<td>11.41</td>
</tr>
<tr>
<td>M0</td>
<td>P-tens.</td>
<td>Des.Pn</td>
<td>Des.Mn</td>
<td>e/h</td>
</tr>
<tr>
<td>148.52</td>
<td>-480.00</td>
<td>350.15</td>
<td>10.47</td>
<td>0.00249</td>
</tr>
</tbody>
</table>

### TRACK 2.0

**TRACK 2.0 generates the following output in addition to the above examples:**

<table>
<thead>
<tr>
<th>P0</th>
<th>Pn,max</th>
<th>Pn</th>
<th>Mn</th>
<th>Pn</th>
<th>Mn</th>
<th>(@ Z )</th>
</tr>
</thead>
<tbody>
<tr>
<td>*</td>
<td>695.93</td>
<td>77.23</td>
<td>347.96</td>
<td>148.53</td>
<td></td>
<td></td>
</tr>
<tr>
<td>*</td>
<td>637.93</td>
<td>93.16</td>
<td>289.97</td>
<td>157.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pn</td>
<td>579.94</td>
<td>107.06</td>
<td>231.98</td>
<td>164.41</td>
<td></td>
<td></td>
</tr>
<tr>
<td>*</td>
<td>521.94</td>
<td>118.23</td>
<td>173.98</td>
<td>170.18</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pn</td>
<td>463.95</td>
<td>129.01</td>
<td>115.99</td>
<td>163.66</td>
<td></td>
<td></td>
</tr>
<tr>
<td>NOMINAL</td>
<td>*</td>
<td>405.96</td>
<td>139.03</td>
<td>57.99</td>
<td>156.37</td>
<td></td>
</tr>
<tr>
<td>AXIAL</td>
<td>*</td>
<td>695.93</td>
<td>77.23</td>
<td>347.96</td>
<td>148.53</td>
<td></td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>*</td>
<td>637.93</td>
<td>93.16</td>
<td>289.97</td>
<td>157.71</td>
<td></td>
</tr>
<tr>
<td>Pb</td>
<td>579.94</td>
<td>107.06</td>
<td>231.98</td>
<td>164.41</td>
<td></td>
<td></td>
</tr>
<tr>
<td>*</td>
<td>521.94</td>
<td>118.23</td>
<td>173.98</td>
<td>170.18</td>
<td></td>
<td></td>
</tr>
<tr>
<td>*</td>
<td>463.95</td>
<td>129.01</td>
<td>115.99</td>
<td>163.66</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P-tens</td>
<td>*</td>
<td>M0</td>
<td>Mn,</td>
<td>Pn</td>
<td>Mn</td>
<td>BENDING</td>
</tr>
<tr>
<td>*</td>
<td>405.96</td>
<td>139.03</td>
<td>57.99</td>
<td>156.37</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### D1.F.5.6.1 Slenderness Effects and Analysis Consideration

Slenderness effects are extremely important in designing compression members. The ACI 318 code specifies two options by which the slenderness effect can be accommodated (Section 10.10 & 10.11 ACI-318). One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed-end moments, the effect of deflections on moments and forces, and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD.Pro uses both these options. To perform the first type of analysis, use the command `PDELTA ANALYSIS` instead of `PERFORM ANALYSIS`. This analysis method will accommodate the requirements as specified in Section 10.10 of the ACI-318 Code, except for the effects of the duration of the loads. It is felt that this effect may be...
safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If it is desired, STAAD.Pro can also accommodate any arbitrary moment magnification factor (second option) as an input, in order to provide some safety due to the effects of the duration of loads.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, moment-magnification is based on empirical formula and assumptions on sidesway.

Considering all this information, it is our belief that a PDELTA ANALYSIS, as performed by STAAD.Pro, is most appropriate for the design of concrete members. However, you should note that to take advantage of this analysis, all combinations of loadings must be provided as repeat load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a repeat load case consists of one or more primary load case which are revised during the PDelta analysis based on the deflections. Also note that you must provide the proper factored loads (e.g., 1.4 for DL etc.). STAAD.Pro does not factor the loads automatically.

Moment Magnification per the 2011 Editions

If the moment magnification method is used in lieu of an exact analysis, columns designed per ACI 318-11 in STAAD.Pro can consider sway using the moment magnification procedure. The second order effects are approximated by magnified moments for slender columns (as determined by clause 10.10.1) in sway and non-sway frames. The moment magnification procedure described in clause 10.10.6 of the code is used for non-sway frame columns and the procedure in clause 10.10.7 is used for sway frame columns.

The parameters BDY, BDZ, SKY, SKZ, SLY, SLZ, SQY, SQZ, SWY, and TRN apply to these calculations.

**Note:** You must set the MMAG parameter to 0 in order to consider sidesway for slender columns. Otherwise, the MMAG value is used directly.

**D1.F.6 Slab Design**

Slab design is performed only for the moments MX and MY at the center of an element. Design will not be performed for SX, SY, SXY, SQX, SQY, or MXY. Also, design is not performed at any other point on the surface of the element.

**Note:** Element design is only supported for ACI 318-08 and earlier editions of the code.

A typical example of element design output is shown below. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement (D1.F.2 Section Types for Concrete Design (on page 1480)). The parameters FYMAIN, FC, and CLEAR listed in D1.F.3 Design Parameters (on page 1481) are relevant to slab design. Other parameters mentioned in Table 3.1 are not applicable to slab design. Please note that the default value of clear cover - parameters CLT and CLB - for plate elements is 0.75 inches, as shown in D1.F.3 Design Parameters (on page 1481).
Element design for flexure is done using the same rules that are used for beam design. A unit width (1 meter) is assumed as the width of the beam. This suits plate elements as MX and MY are in units of moment per unit width. Reinforcement is reported in units of mm² per unit width. Longitudinal direction corresponds to the direction of reinforcement for MX, transverse corresponds to that for MY. Longitudinal reinforcement is the outer layer of reinforcement and transverse is the inner layer.

Elements are not designed for shear forces or axial stress.

**Example Element Design Output**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LONG. REINF</th>
<th>MOM-X /LOAD</th>
<th>TRANS. REINF</th>
<th>MOM-Y /LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(SQ.IN/FT)</td>
<td>(K-FT/FT)</td>
<td>(SQ.IN/FT)</td>
<td>(K-FT/FT)</td>
</tr>
<tr>
<td>FY</td>
<td>60.000 KSI</td>
<td>4.000 KSI</td>
<td>0.750 IN</td>
<td>0.750 IN</td>
</tr>
<tr>
<td>COVER (TOP):</td>
<td>0.750 IN</td>
<td>TH: 6.000 IN</td>
<td></td>
<td></td>
</tr>
<tr>
<td>47 TOP : Longitudinal direction - Only minimum steel required.</td>
<td>0.130</td>
<td>0.00 / 0</td>
<td>0.130</td>
<td>0.00 / 0</td>
</tr>
<tr>
<td>47 TOP : Transverse direction - Only minimum steel required.</td>
<td>0.562</td>
<td>11.60 / 1</td>
<td>0.851</td>
<td>14.83 / 1</td>
</tr>
</tbody>
</table>

**Example**

Element design per the ACI 318-2008 code

UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2008
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
Example
Element design per the ACI 318-1999 code
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN

D1.F.6.1 Slabs and RC Designer
The D. Interactive Concrete Design (on page 1005) is used for slab design.
Contact the http://www.bentley.com/serviceticketmanager for further information.

D1.F.7 Scope of ACI 318 Code Implemented
The following describes the code clauses from ACI 318-14 implemented for beams and columns.

Note: Plate elements and deep beams cannot be designed per ACI 318-14. Use an older edition of the ACI code to design these elements if needed.

Materials
- Beams
  19.2.2
  19.2.3.1
  19.2.4.2
- Columns
  19.2.2
  19.2.4.2

Minimum Reinforcement
- Beams
  9.6.1.1
  9.6.1.2
  9.6.1.3
  9.7.2.1
  9.7.6.4.2
  9.7.6.4.3
- Columns
  10.6.1
Clauses 9.7.6.4.2 and 9.7.6.4.3 are not included in the standard design criteria of the minimum reinforcement since the design of these criteria should consider the rebar including the bar detailing and thus they should be applied in a second design after performing the strength design and the bar detailing.

Clause 9.6.1.3 is also not included in the standard criteria since the reduction of the minimum reinforcement is optional.

**Service**
- Beams
  - 9.7.2.2
  - 9.7.2.3

**Strength**
- Beams
  - 9.5
  - 9.5.1
  - 9.5.2
  - 9.5.3
  - 9.5.4
  - 9.6.3.1
  - 9.6.3.3
  - 9.6.4.2
  - 9.7.5.2
  - 9.7.6.2.2
  - 9.7.6.3.1
  - 9.7.6.3.3
  - 9.7.3.3
  - 9.6.4.3
  - 20.2.2.4a
  - 20.2.2.1
  - 20.2.2.2
  - 21.2.1
  - 22.2.2.2
  - 22.5.5
  - 22.2.2.3
  - 22.7.4
  - 22.5.1.2
  - 22.7.6.1
  - 22.7.7.1
Design
D. Design Codes

- Columns
  10.5
  10.5.1
  10.5.2
  10.5.3
  10.6.2
  10.7.6.5.2
  20.2.2.4a
  20.2.2.1
  20.2.2.2
  22.5
  22.2.2.2
  22.5.5
  22.2.2.3
  22.7.4
  22.5.1.2
  22.7.6.1

Ductility
- Beams
  9.3.3.1

Development Length
- Beams
  25.4.1.4
  25.4.2
  25.4.3
- Columns
  25.4.1.4
  25.4.2
  25.4.3

Span Detailing
- Beams
  9.7.3.3
  9.7.3.8
  9.7.7

D1.G. American Codes - Timber Design per AITC Code
D1.G.1 Design Operations

STAAD.Pro supports timber design per two versions of the AITC code: 1985 and 1994. The implementation of both the codes is explained below.

To access the 1994 edition, specify the commands:

```
CODE AITC

or

CODE AITC 1994
```

To access the 1984 edition, specify the commands:

```
CODE AITC 1984

or

CODE TIMBER
```

D1.G.1.1 1994 AITC Code Implementation

The salient aspects of design in accordance with the 4th edition (1994) of the *Timber Construction Manual* published by the American Institute of Timber Construction are:

1. Design can be performed for two types of timber sections: dimensional timber sections (i.e., sawn lumber) and glulaminated sections.

2. The program includes a database of dimensional timber sections with this code.

Implementation of Dimensional Lumber Properties

The database of sawn lumber sections, listed in Table 8.1 of the 1994 AITC Manual, is implemented in the program. Some of the key aspects of this implementation are:

In the property tables in the AITC manual, one will find that, for any particular species of timber, the Modulus of Elasticity (E) and allowable stresses may vary with the cross-section size. For example, a 2x4 Douglas Fir-Larch, Select Structural member has an E of 1900 ksi and an allowable bending stress, $F_b$, of 1450 psi. A 5x5 Douglas Fir-Larch, Select Structural, Beam or Stringer member has an E of 1600 ksi and an allowable bending stress, $F_b$, of 1600 psi. And a 5x5 Douglas Fir-Larch, Select Structural, Post or Timbers member has an E of 1600 ksi and an allowable bending stress, $F_b$, of 1750 psi.

So, in the STAAD timber database for sawn lumber, for each species and grade of timber, the section size, or properties are associated with the Modulus of Elasticity and allowable stresses for the cross-section. When a section is assigned, its E and allowable stresses are automatically fetched along with its properties. The material properties of Southern Pine members were taken from Table 8.4 of the 1994 AITC manual. For all other species with section sizes 2"-4" wide, the material properties have been taken from Table 8.3. For all non-Southern Pine species with section sizes greater than 5"x5", the material properties are obtained from Table 8.6 of the 1994 AITC Manual.

Please note that not all section sizes listed in Table 8.1 are available in every species. Some sizes are not produced for particular species. For example, the Aspen species only produces sizes from 2"-4" wide. It does not produce sizes 5"x5" and larger. This can be observed by comparing Table 8.3, where Aspen is listed as an available species, to Table 8.6, where Aspen is not listed as an available species. Also note that although 1" wide members are listed in Table 8.1, there are no values available in the species properties tables; Table 8.4, Table 8.5, and Table 8.6. AITC does not allow for the structural design of these small members.
D1.G.1.2 1985 AITC Code Implementation

STAAD’s Timber design module per the 1985 AITC code (Timber Construction Manual, 3rd. Edition, 1985) allows design of Glulam timber sections. It also conforms to the National Design Specification for Wood Construction and Supplement (NDS) and building codes like Uniform Building Code (UBC), Basic/National Building Code and Standard Building Code. Some of the main features of the program are:

1. This feature is for Glulam Timber only (design of dimensional lumber are available for 1994 AITC only).
2. Code check and design of members as per TCM - AITC.
3. Design values for Structural Glued Laminated Timber tables are in-built into the program. The program accepts Table no., Combination and Species specifications as inputs (e.g., 1:16F-V3-SP/SP) and reads design values from in-built tables.
4. Incorporates all the following Allowable stress modifiers:
   i. Duration of Load Factor
   ii. Size Factor
   iii. Form Factor
   iv. Lateral stability of Beams and Columns
   v. Moisture Content Factor
   vi. Temperature and Curvature factors.

   The allowable stresses for bending, tension, compression, shear and Moduli of elasticities are modified accordingly.
5. Determines slenderness for beams and columns (Short, intermediate and long) and checks for min. eccentricity, lateral stability, buckling, bending and compression, bending and tension and horizontal shear against both axes.
6. The output results show sections provided or chosen, actual and allowable stresses, governing condition and ratios of interaction formulae and the relevant AITC clause nos. etc for each individual member.

Related Links
- TR.52 Timber Design Specifications (on page 2857)
- TR.52.1 Timber Design Parameter Specifications (on page 2857)

D1.G.2 Allowable Stress per AITC Code

Explanation of terms and symbols used in this section

Table 105: Timber design nomenclature

<table>
<thead>
<tr>
<th>Symbols</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_a$</td>
<td>Actual compression or tension stress (in PSI). For tension, the axial load is divided by net sectional area (i.e, NSF x X-area).</td>
</tr>
<tr>
<td>FA</td>
<td>Allowable design value for compression or tension (in PSI) modified with applicable modifiers or calculated based on slenderness in case of compression.</td>
</tr>
<tr>
<td>$f_{bz}$</td>
<td>Actual bending stresses about local Z and Y axis (in PSI).</td>
</tr>
</tbody>
</table>
Symbols | Description
--- | ---
FBZ | Allowable design values for bending stresses about local Z and Y axis (in PSI) modified by the applicable modifiers.
FBY | 
JZ | Modifier for P-DELTA effect about the Z and Y axis respectively as explained in formula 5-18 of TCM.
JY | 
$f_{vz}$ | Actual horizontal shear stresses.
$f_{vy}$ | 
FVZ | Allowable horizontal shear stresses.
FVV | 
VZ | Shear in local Z and local Y direction.
VY | 
ZD | Depth of section in local Z and Y axis.
YD | 
EZ | Minimum eccentricity along Z and Y axis.
EY | 
CFZ | Values of the size factors in the z-axis and y-axis, respectively.
CFY | 
CLZ | Represent the factors of lateral stability for beams about the z-axis and y-axis, respectively.
CLY | 
RATIO | Permissible ratio of stresses. The default value is 1.0.

**D1.G.2.1 Combined Bending and Axial Tension**

The following interaction formulae are checked:

\[
\frac{f_a}{FA} + \frac{f_{bz}}{(FBZ \times CFZ)} + \frac{f_{by}}{(FBY \times CFY)} \leq
\]

Lateral stability check with Net compressive stress:

\[
\frac{f_a}{FA} + \frac{f_{bz}}{(FBZ \times CLZ)} + \frac{f_{by}}{(FBY \times CLY)} \leq
\]

**D1.G.2.2 Combined Bending and Axial Compression**

\[
\frac{f_a}{FA} + \frac{f_{bz}}{(FBZ-JZ \times f_a)} + \frac{f_{by}}{(FBY-JY \times f_a)} \leq
\]

Applicability of the size factor:
D1.G.2.3 Minimum Eccentricity

The program checks against min. eccentricity in following cases:

a. The member is a FRAME member and not a truss member and under compression.

b. The value of actual axial compressive stress does not exceed 30% of the allowable compressive stress.

c. The actual moments about both axes are less than moments that would be caused due to min. eccentricity. In this approach, the moment due to min. eccentricity is taken as the compressive load times an eccentricity of 1 in. or 0.1 x depth whichever is larger.

In case of min. eccentricity,

\[ f_{bz} = f_a \times \frac{6+1.5 \times JZ}{EZ/ZD} \]
\[ f_{by} = f_a \times \frac{6+1.5 \times JY}{EY/YD} \]

the following conditions are checked:

\[ f_a/FA + f_{bz}/(FBZ-JZ \times f_a) \leq \]
\[ f_a/FA + f_{by}/(FBY-JY \times f_a) \leq \]

D1.G.2.4 Shear Stresses

Horizontal stresses are calculated and checked against allowable values:

\[ f_{vz} = 3 \times VY / (2 \times \text{Area} \times \text{NSF}) \leq FVZ \]
\[ f_{vy} = 3 \times VZ / (2 \times \text{Area} \times \text{NSF}) \leq FVY \]

Related Links

- TR.52 Timber Design Specifications (on page 2857)
- TR.52.1 Timber Design Parameter Specifications (on page 2857)

D1.G.3 Input Specification

A typical set of input commands for STAAD.Pro timber design per AITC 1984 is listed below:

<table>
<thead>
<tr>
<th>UNIT KIP INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>PARAMETER</td>
</tr>
<tr>
<td>CODE TIMBER</td>
</tr>
<tr>
<td>GLULAM 1:16F-V3-DF/DF MEMB 1 TO 14</td>
</tr>
<tr>
<td>GLULAM 1:24F-V5-SP/SP MEMB 15 TO 31</td>
</tr>
<tr>
<td>GLULAM 20F-V1-DF/WW MEMB 32 TO 41</td>
</tr>
<tr>
<td>LAMIN 1.375 LY 168.0 MEMB 5 9 15 TO 31</td>
</tr>
<tr>
<td>LZ 176.0 MEMB 1 TO 4 6 7 8 10 TO 14</td>
</tr>
<tr>
<td>LUZ 322.6 ALL</td>
</tr>
<tr>
<td>LUY 322.6 ALL</td>
</tr>
</tbody>
</table>
D1.G.3.1 Explanation of Input Commands and Parameters

Specify PARAMETER and then CODE TIMBER to start TIMBER DESIGN before specifying the input parameters. The user must provide the timber grade (GLULAM GRADE) for each member he intends to design. The parameters can be specified for all or specified list of members. If a parameter is not specified, the default value is assigned to it. See following INPUT PARAMETERS LIST TABLE for description and default values of the parameters.

D1.G.3.2 Glulam Grade and Allowable Stresses from Table

The allowable stresses for GLULAM members are read in from Table-1 and Table-2 of AITC for design values for Structural Glued Laminated Timber. The structural members are to be specified in the following manner:

![Figure 168: AITC Table-1 glulam members](image1)

![Figure 169: AITC Table-2 glulam members](image2)

For TABLE-2 members, the applicable stress values are selected based on the depth and the number of laminations.

**Note:** The lamination thickness (in inches) can be specified. Typical values are 1 3/8 inches or 1 1/2 inches. If not specified, a default value of 1 1/2 inches assumed by the program.

**Related Links**
- TR.52 Timber Design Specifications (on page 2857)
- TR.52.1 Timber Design Parameter Specifications (on page 2857)
D1.G.4 Naming Conventions for Sections

The following conventions are used to describe timber sections in STAAD.Pro

D1.G.4.1 Dimensional Lumber sections

As can be seen from Tables 8.3 through 8.6 of the AITC 1994 manual, one or more of the following attributes have to be considered while choosing a section:

- Species
- Commercial Grade
- Size classification
- Nominal size of the section
- Grading rules agency

STAAD uses a naming convention that incorporates all of the above. Shown below is the name of a section that has characteristics as shown. It may be found on page 8-637 of the AITC 1994 manual.

Species: Douglas Fir Larch
Commercial Grade: Dense Select Structural
Size Classification: Beams
Nominal size: 5”x 5”
Grading Rules Agency: WCLIB

![Figure 170: Timber naming conventions](image)

Implementation of Glulam Properties

For Glulam sections, each glulam designation has an associated value of Modulus of Elasticity and a set of allowable stresses. However, these values are not dependent on the size of the cross section. For example, a 3-1/8” x 6” 24F-V8 DF/DF beam and a 6-3/4” x 30” 24F-V8 DF/DF beam both have an E of 1,600 ksi and an allowable bending stress in the tension zone, $F_{bx}$, of 2,400 psi.

Therefore, in STAAD’s glulam database, the section sizes are not linked to the glulam type. Users may specify any cross-section size they choose and pick the desired glulam type. The Modulus of Elasticity and allowable stresses associated with that glulam are assigned to the member. The material properties for the Glulam database are taken from Table 1 of AITC 117-93 – Design Standard Specifications for Structural Glued Laminated Timber of Softwood Species. This publication has been reproduced in the AITC 1994 manual starting from page 8-843.

Example for Dimensional Timber

```
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC DFLN_SS_4X4
E 273600
```
D1.G.4.2 Glulam sections

The STAAD name for glulam sections incorporates

- Combination Symbol
- Species-Outer Laminations/Core Laminations

Shown here is a typical section listed in page 8-854 of the AITC manual.

GLT-24F-V11_DF/DFS

Implementation of Material Constants

As explained in the previous paragraphs, for sawn lumber as well as glulam sections, E is built into the database and gets automatically assigned to the member along with the section dimensions. Density, Poisson’s ratio and Alpha (coefficient of thermal expansion) have to be assigned separately. If they are not assigned, the analysis engine will use default values for those.

Example for Glulam Timber:

UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_WET_DF/DF
E 191923
POISSON 0.15
DENSITY 0.025
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
8 PRIS YD 1.5 ZD 0.427083
CONSTANTS
MATERIAL GLT-24F-V8_WET_DF/DF MEMB 8

D1.G.5 Design Parameters

The timber design parameters for the AITC codes.
### D1.G.5.1 AITC 1994 Parameters

#### Table 106: AITC 1994 Timber Design Properties

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>AITC 1994 Code</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>STAAD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CB</td>
<td>$C_b$</td>
<td>1.0</td>
<td>Bearing Area Factor, Table 4.13</td>
</tr>
<tr>
<td>CFB</td>
<td>$C_F$</td>
<td>1.0</td>
<td>Size Factor for Allowable Bending Stress, see Table 8.3, 8.4, 8.5, 8.6, 8.7</td>
</tr>
<tr>
<td>CPC</td>
<td>$C_F$</td>
<td>1.0</td>
<td>Size Factor for Allowable Compression Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7</td>
</tr>
<tr>
<td>CFT</td>
<td>$C_F$</td>
<td>1.0</td>
<td>Size Factor for Allowable Tension Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7</td>
</tr>
<tr>
<td>CFU</td>
<td>$C_{fu}$</td>
<td>1.0</td>
<td>Flat Use Factor, see Table 4.9</td>
</tr>
<tr>
<td>CSS</td>
<td>$C_H$</td>
<td>1.0</td>
<td>Shear Stress Factor, Section 4.5.14</td>
</tr>
<tr>
<td>CMB</td>
<td>$C_M$</td>
<td>1.0</td>
<td>Wet service Factor for Allowable Bending Stress, see Table 4.8</td>
</tr>
<tr>
<td>CMC</td>
<td>$C_M$</td>
<td>1.0</td>
<td>Wet service Factor for Allowable Compression Parallel to Grain, see Table 4.8</td>
</tr>
<tr>
<td>CME</td>
<td>$C_M$</td>
<td>1.0</td>
<td>Wet service Factor for Modulus of Elasticity, see Table 4.8</td>
</tr>
<tr>
<td>CMP</td>
<td>$C_M$</td>
<td>1.0</td>
<td>Wet service Factor for Allowable Compression Perpendicular to Grain, see Table 4.8</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
<td></td>
</tr>
<tr>
<td>STAAD AITC 1994 Code</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CMT</td>
<td>CM</td>
<td>1.0</td>
<td>Wet service Factor for Allowable Tension Parallel to Grain, see Table 4.8</td>
</tr>
<tr>
<td>CMV</td>
<td>CM</td>
<td>1.0</td>
<td>Wet service Factor for Allowable Shear Stress Parallel to Grain, see Table 4.8</td>
</tr>
<tr>
<td>CR</td>
<td>Cr</td>
<td>1.0</td>
<td>Repetitive Member Factor, see Section 4.5.10</td>
</tr>
<tr>
<td>CSF</td>
<td>CF</td>
<td>1.0</td>
<td>Form Factor, see Section 4.5.12</td>
</tr>
<tr>
<td>CTM</td>
<td>Ct</td>
<td>1.0</td>
<td>Temperature Factor, see Table 4.11</td>
</tr>
<tr>
<td>CTT</td>
<td>CT</td>
<td>1.0</td>
<td>Buckling Stiffness Factor, see Section 4.5.15</td>
</tr>
<tr>
<td>KB</td>
<td>Kb</td>
<td>1.0</td>
<td>Buckling Length Coefficient to calculate Effective Length</td>
</tr>
<tr>
<td>KBD</td>
<td>Kbd</td>
<td>1.0</td>
<td>Buckling Length Coefficient for Depth to calculate Effective Length</td>
</tr>
<tr>
<td>KBE</td>
<td>KeE</td>
<td>0.609</td>
<td>Euler Buckling Coefficient for Beams, see Section 5.4.11</td>
</tr>
<tr>
<td>KCE</td>
<td>KeE</td>
<td>1.0</td>
<td>Euler Buckling Coefficient for Columns, see Section 5.8.2</td>
</tr>
<tr>
<td>KEY</td>
<td>Key</td>
<td>1.0</td>
<td>Buckling Length Coefficient in Y Direction</td>
</tr>
<tr>
<td>KEZ</td>
<td>KeZ</td>
<td>1.0</td>
<td>Buckling Length Coefficient in Z Direction</td>
</tr>
<tr>
<td>KL</td>
<td>Kl</td>
<td>1.0</td>
<td>Load Condition Coefficient, Table 4.10</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
<td></td>
</tr>
<tr>
<td>STAAD AITC 1994 Code</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LZ</td>
<td>LZ</td>
<td>Member Length</td>
<td>Effective shear length in the z direction for Column Stability Check, Le=Ke*L</td>
</tr>
<tr>
<td>LY</td>
<td>LY</td>
<td>Member Length</td>
<td>Effective shear length in the y direction for Column Stability Check, Le=Ke*L</td>
</tr>
<tr>
<td>LUZ</td>
<td>LUZ</td>
<td>Member Length</td>
<td>Member length in the z direction for Beam Stability Check, Lu=Kb<em>l +Kbd</em>d</td>
</tr>
<tr>
<td>LUY</td>
<td>LUY</td>
<td>Member Length</td>
<td>Member length in the y direction for Beam Stability Check, Lu=Kb<em>l +Kbd</em>d</td>
</tr>
<tr>
<td>CDT</td>
<td>CDT</td>
<td>1.0</td>
<td>Load Duration Factor</td>
</tr>
<tr>
<td>CCR</td>
<td>CCR</td>
<td>1.0</td>
<td>Curvature factor (Section 4.5.11)</td>
</tr>
<tr>
<td>INDEX</td>
<td>INDEX</td>
<td>10</td>
<td>Exponent value in the Volume Factor Equation (Section 4.5.6)</td>
</tr>
<tr>
<td>CV</td>
<td>CV</td>
<td>1.0</td>
<td>Volume Factor (Section 4.5.6)</td>
</tr>
<tr>
<td>CC</td>
<td>CC</td>
<td>0.8</td>
<td>Variable in Column Stability Factor, Cp (Section 5.8.2, Eqn 5-14)</td>
</tr>
<tr>
<td>SRC</td>
<td>SRC</td>
<td>1.0</td>
<td>Slenderness ratio of Compression member</td>
</tr>
<tr>
<td>SRT</td>
<td>SRT</td>
<td>1.0</td>
<td>Slenderness ratio of Tension member</td>
</tr>
<tr>
<td>RATIO</td>
<td></td>
<td>1.0</td>
<td>Permissible ratio of actual to allowable stress</td>
</tr>
</tbody>
</table>
### Parameter Name | Default Value | Description |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>STAAD AITC 1994 Code</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = design at start and end nodes and those locations specified by the SECTION command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes. Note: See D1.A.6 Design Parameters (on page 1378).</td>
</tr>
</tbody>
</table>

### D1.G.5.2 AITC 1984 Parameters

Table 107: AITC 1985 Timber Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = design at start and end nodes and those locations specified by the SECTION command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes. Note: See D1.A.6 Design Parameters (on page 1378).</td>
</tr>
<tr>
<td>CCR</td>
<td>1.0</td>
<td>Curvature factor</td>
</tr>
<tr>
<td>CDT</td>
<td>1.0</td>
<td>Duration of load factor</td>
</tr>
<tr>
<td>CSF</td>
<td>1.0</td>
<td>Form factor</td>
</tr>
<tr>
<td>CTM</td>
<td>1.0</td>
<td>Temp. factor</td>
</tr>
<tr>
<td>LAMINATION</td>
<td>1.50 inch</td>
<td>Thickness of lamination in inch (1.50 or 1.375)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LUZ</td>
<td>1.92*L</td>
<td>Unsupported effective length for beam in z.</td>
</tr>
<tr>
<td>LUY</td>
<td>1.92*L</td>
<td>Unsupported effective length for beam in y.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Same as above in y-axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Effective length of the column in z-axis.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members. (both shear and tension stresses are based on sectional area ( \times ) nsf)</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual to allowable stresses.</td>
</tr>
</tbody>
</table>
| WET           | 0.0           | 0.0 - dry condition  
|               |               | 1.0 - wet condition  
|               |               | wet use factors are in-built |

### D1.G.6 Member Design Capabilities

STAAD.Pro is capable of performing member design functions for both 1984 and 1994 editions of AITC.

**Tip:** The User Interface can be used to easily assign design commands to members.

#### D1.G.6.1 Code Checking

The `CHECK CODE` command enables the user to check the adequacy of the size \( (YD \times ZD) \) provided in the `MEMBER PROPERTIES` for the most critical forces and moments. The program prints whether the member has passed or failed, the critical conditions and the value of the ratio.

#### D1.G.6.2 Member Selection

Member selection is limited to AITC 1994.

The `SELECT MEMBER` command starts with the min. permissible depth (or min. depth provided thru `DMIN` parameter) and checks the code. If the member fails with this depth, the thickness is increased by one lamination thickness and the code requirements are checked again. The process is continued till the section passes all the code requirements. This ensures the least weight section for the member. If the depth of the section reaches max. allowable or available depth and the member still fails, you can have the following options for redesign:

- **a.** Change the width or increase the max. allowable depth (\( \text{DMAX} \))
- **b.** Change the timber grade
- **c.** Change the design parameters
D1.G.7 Orientation of Lamination

Laminations are always assumed to lie along the local Z-plane of the member. In the MEMBER PROPERTIES section, YD always represents the depth of the section across the grain and ZD represents the width along the grain. This is in accordance with the sign convention conforming to SET Y UP (default).

![Figure 171: Orientation of lamination](image)

D1.G.8 Tabulated Results of Member Design

For CODE CHECKING and/or MEMBER SELECTION the output results are printed as shown in the previous section. The items are explained as follows:

- **MEMBER** refers to the member number for which the design is performed.
- **TABLE** refers to the size of the PRISMATIC section (B X D or ZD X YD).
- **RESULT** prints whether the member has PASSED or FAILED.
- **CRITICAL COND** refers to the CLAUSE or FORMULA NO. from the TIMBER CONSTRUCTION MANUAL (3rd Edition, AITC-1985) which governed the design. See following table:

<table>
<thead>
<tr>
<th>Critical Condition</th>
<th>Governing Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause 5-19</td>
<td>Axial Compression and Bending with MINIMUM ECCENTRICITY.</td>
</tr>
<tr>
<td>Clause 5-18</td>
<td>Axial Compression and Bending</td>
</tr>
<tr>
<td>Clause 5-42</td>
<td>Axial Tension and Bending</td>
</tr>
<tr>
<td>Clause 5-24</td>
<td>Horizontal Shear</td>
</tr>
<tr>
<td>Clause 5.40</td>
<td>Lateral stability for net compressive stress in case of Tension and Bending.</td>
</tr>
</tbody>
</table>

- **RATIO** prints the ratio of the actual stresses to allowable stresses for the critical condition. This ratio is usually the cumulative ratio of stresses in the interaction formula. In case of shear governing the design, it
means the ratio of the actual shear stress to allowable shear stress. If this value exceeds the allowable ratio (default 1.0) the member is FAILed.

f. LOADING provides the load case number that governed.

g. FX, MY and MZ provide the design axial force, moment in local Y axes and moment in local Z axes respectively. FX value is followed by a letter C or T to denote COMPRESSION or TENSION.

h. LOCATION specifies the actual distance from the start of the member to the section where design forces govern in case BEAM command or SECTION command is specified.

OUTPUT parameters that appear within the box are explained as follows:

a. MEMB refers to the same member number for which the design is performed.

b. GLULAM GRADE refers to the grade of the timber.

c. LAM refers to lamination thickness provided in the input or assumed by the program. See INPUT PARAMETERS section.

d. LZ, LY, LUZ and LUY are the effective lengths as provided or calculated. See INPUT PARAMETERS section.

e. JZ and JY are the modifiers for the P-DELTA effect about Z-axis and Y-axis respectively. These are calculated by the program.

f. CDT, CSF, WET, CCR, CTM are the allowable stress modifiers explained in the INPUT PARAMETERS section.

g. CFZ and CFY are values of the size factors in the Z-axis and Y-axis respectively. CLZ and CLY represent the factors of lateral stability for beams about Z-axis and Y-axis respectively. These values are printed to help the user see the intermediate design values and re-check the design calculations.

h. \( f_{ax}, f_{bz}, f_{by}, f_{vz}, \) and \( f_{vy} \) are the actual axial stress, bending stresses about Z and Y axes and horizontal shear stresses about Z and Y axes respectively. If the bending moments about both axes are less then the eccentric moments based on min. eccentricity then bending stresses are calculated based on the min. eccentricity. Refer DESIGN OPERATIONS section for details.

i. FA, FBZ, FBY, FVZ, and FVY are the final allowable axial, bending (Z and Y axes) and horizontal shear (Z and Y axes) stresses. See D1.G.2 Allowable Stress per AITC Code (on page 1528) for details.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>10.750X16.500 GLULAM GRADE:GLT-24F-V8_DF/DF</td>
<td>PASS</td>
<td>CL.5.9.2</td>
<td>0.014</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>4583.17 C</td>
<td>0.00</td>
<td>1310.87</td>
<td>12.0000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>LEZ = 144.000</td>
<td>LEY = 144.000</td>
<td>LUZ = 144.000</td>
<td>LUY = 144.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>CD = 1.000</td>
<td>CMB = 1.000</td>
<td>CMT = 1.000</td>
<td>CMC = 1.000</td>
<td>CMP = 1.000</td>
</tr>
<tr>
<td></td>
<td>CMV = 1.000</td>
<td>CME = 1.000</td>
<td>CFB = 1.000</td>
<td>CFT = 1.000</td>
<td>CFC = 1.000</td>
</tr>
<tr>
<td></td>
<td>CFU = 1.000</td>
<td>CR = 1.000</td>
<td>CTT = 1.000</td>
<td>CC = 1.000</td>
<td>CF = 1.000</td>
</tr>
<tr>
<td></td>
<td>CT = 1.000</td>
<td>CH = 1.000</td>
<td>CB = 1.000</td>
<td>CI = 1.000</td>
<td>CV = 0.000</td>
</tr>
<tr>
<td></td>
<td>CLY = 0.999</td>
<td>CLZ = 0.997</td>
<td>CP = 0.934</td>
<td>c = 0.000</td>
<td>E' = 1600000.122 PSI</td>
</tr>
<tr>
<td></td>
<td>ACTUAL STRESSES : (POUND INCH)</td>
<td>fc = 25.839 ft = 0.000</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
D1.G.9 Examples

The following conventions are used to describe timber sections in STAAD.Pro

**D1.G.9.1 Example for dimensional lumber**

**STAAD PLANE EXAMPLE FOR DIMENSIONAL LUMBER**

UNIT FEET POUND

JOINT COORDINATES
1 0 0 0; 2 6 0 0; 3 12 0 0; 4 18 0 0; 5 24 0 0; 6 6 3 0; 7 12 6 0; 8 18 3 0;

MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 1 6; 6 6 7; 7 7 8; 8 8 5; 9 2 6; 10 3 7; 11 4 8; 12 6 3; 13 3 8;

UNIT FEET POUND

DEFINE MATERIAL START
ISO orthotropic DFLR_SS_2X4
E 2.736e+008
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006
ISO orthotropic DFLR_SS_3X6
E 2.736e+008
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006

END DEFINE MATERIAL

MEMBER PROPERTY AITC
1 TO 4 9 TO 11 TABLE ST DFLR_SS_2X4
5 TO 8 12 13 TABLE ST DFLR_SS_3X6

CONSTANTS
MATERIAL DFLR_SS_2X4 MEMB 1 TO 4 9 TO 11
MATERIAL DFLR_SS_3X6 MEMB 5 TO 8 12 13

MEMBER RELEASE
9 TO 13 START MP 0.99
9 TO 13 END MP 0.99
6 END MP 0.99
7 START MP 0.99

SUPPORTS
1 PINNED
5 FIXED BUT FX MZ

UNIT FEET POUND

LOAD 1 DEAD+LIVE LOAD
SELFWEIGHT Y -1

MEMBER LOAD
"Design

D. Design Codes

1 TO 4 UNI GY -30
5 TO 8 UNI GY -40
LOAD 2 SNOW LOAD
MEMBER LOAD
5 TO 8 UNI GY -50
LOAD 3 WIND LOAD
MEMBER LOAD
5 6 UNI Y -30
7 8 UNI Y 25
LOAD COMB 11 D+L+SNOW
1 1.0 2 1.0
LOAD COMB 12 D+L+SNOW+WIND
1 1.0 2 1.0 3 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
*Design per AITC
PARAMETER
CODE AITC
BEAM 1.0 ALL
CHECK CODE ALL
FINISH

D1.G.9.2 Example for Glulaminated lumber

STAAD PLANE EXAMPLE FOR GLULAM DESIGN
INPUT WIDTH 79
UNIT FEET KIP
JOIN COORDINATES
1 0 0 0; 2 12 0 0; 3 24 0 0; 4 36 0 0; 5 0 12 0; 6 6 10 0; 7 18 6 0; 8 30 2 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 8 4; 8 1 5; 9 2 6; 10 3 7; 11 1 6;
12 2 7; 13 3 8;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_DF/DF
E 1600
POISSON 0.15
DENSITY 1.44676e-005
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY
1 TO 7 PRIS YD 16.5 ZD 10.75
8 TO 13 PRIS YD 10.5 ZD 8.75
CONSTANTS
MATERIAL GLT-24F-V8_DF/DF MEMB 1 TO 13
SUPPORTS
1 4 PINNED
UNIT POUND FEET
LOAD 1 DEAD
SELFWEIGHT Y -1
LOAD 2 LIVE
MEMBER LOAD
1 TO 3 UNI GY -100
4 TO 7 UNI GY -100
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER
D1.H. American Codes - Aluminum Design per 1994 ADM


D1.H.1 Member Properties

In order to do this design in STAAD.Pro, the members in the structure must have their properties specified from Section VI of the above-mentioned manual. The section names are mentioned in Tables 5 through 28 of that manual. All of those tables except Table 10 (Wing Channels) and Table 20 (Bulb Angles) are available in STAAD.Pro.

Described below is the command specification for various sections:

D1.H.1.1 Standard single section

```
memb-list  TA ST section-name
```

Example

```
1 5 TA ST CS12X11.8
9 TA ST I8.00X13.1
11 45 67 TA ST LS8.00X8.00X0.625
18 TA ST 1.50PipeX160
15 TA ST T(A-N)6.00X8.00X11.2
23 25 29 TA ST 20X12RectX.500Wall
```

D1.H.1.2 Double channel back-to-back

```
memb-list  TA BACK section-name  SPACING value
```

Example

```
3 TA BACK C(A-N)7X3.61 SPACING 1.5
5 TA BACK C15X17.33 SP 0.75
```

D1.H.1.3 Double channel front-to-front

```
memb-list  TA FRONT section-name  SPACING value
```

Example

```
2 TA FRONT CS12X10.3 SP 1.0
4 TA FR CS10X10.1 SP 0.5
```

D1.H.1.4 Double angle long leg back-to-back

```
memb-list  TA LD section-name  SPACING value
```
Example
14 TA LD LS4.00X3.00X0.375 SP 1.5

**D1.H.1.5 Double angle short leg back-to-back**

memb-list  TA SD section-name  SPACING value

Example
12 TA SD L3.5X3X0.5 SP 0.25
13 TA SD L8X6X0.75 SP 1.0

**D1.H.2 Design Procedure**

The design is done according to the rules specified in Sections 4.1, 4.2 and 4.4 on pages I-A-41 and I-A-42 of the Aluminum code. The allowable stresses for the various sections are computed according to the equations shown in Section 3.4.1 through 3.4.21 on pages I-A-27 through I-A-40. The adequacy of the member is checked by calculating the value of the left-hand side of equations 4.1.1-1, 4.1.1-2, 4.1.1-3, 4.1.2-1, 4.4-1 and 4.4-2. This left-hand side value is termed as RATIO. If the highest RATIO among these equations turns out to be less than or equal to 1.0, the member is declared as having PAssed. If it exceeds 1.0, the member has FAILed the design requirements.

**Note:** The check for torsion per Clause 4.3 for open sections is currently not implemented in STAAD.Pro.

**D1.H.3 Design Parameters**

The following are the parameters for specifying the values for variables associated with the design.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

**Table 109: Aluminum Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as ALUMINUM Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>ALCLAD</td>
<td>0</td>
<td>Defines if material is Alclad. 0 - Material used in the section is not an Alclad. 1 - Material used in the section is an Alclad.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>ALLOY</td>
<td>34</td>
<td>This variable can take on a value from 1 through 40. The default value represents the alloy 6061-T6. See Table 14A.2 below for a list of values for this parameter and the alloy they represent. Table 3.3-1 in Section I-B of the Aluminum specifications provides information on the properties of the various alloys.</td>
</tr>
<tr>
<td>BEAM</td>
<td>0.0</td>
<td>If this parameter is set to 1.0, the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the BEAM value is 0.0, the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (See STAAD manual for details). If neither the BEAM parameter nor any SECTION command is specified, STAAD will terminate the run and ask the user to provide one of those 2 commands. This rule is not enforced for TRUSS members.</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000 in.</td>
<td>Maximum depth permissible for the section during member selection. This value must be provided in the current units.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in</td>
<td>Minimum depth required for the section during member selection. This value must be provided in the current units.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td>Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the allowable stress in axial compression. See Equation 3.4.7.2-6 on page I-A-28 of the Aluminum specifications for details.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor for overall column buckling in the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor for overall column buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.</td>
</tr>
<tr>
<td>LT</td>
<td>Member length</td>
<td>Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the allowable stress in axial compression. See Equation 3.4.7.2-6 on page I-A-28 of the Aluminum specifications for details.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Effective length for overall column buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Effective length for overall column buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.</td>
</tr>
<tr>
<td>PRODUCT</td>
<td>1</td>
<td>This variable can take on a value from 1 through 4. They represent: 1 - All 2 - Extrusions 3 - Drawn Tube 4 - Pipe The default value stands for All. The PRODUCT parameter finds mention in Table 3.3-1 in Section I-B of the Aluminum specifications.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SSY</td>
<td>0.0</td>
<td>Factor that indicates whether or not the structure is subjected to sidesway along the local Y axis of the member. The values are: 0 - Sidesway is present along the local Y-axis of the member 1 - There is no sidesway along the local Y-axis of the member. The sidesway condition is used to determine the value of $C_m$ explained in Section 4.1.1, page I-A-41 of the Aluminum specifications.</td>
</tr>
<tr>
<td>SSZ</td>
<td>0.0</td>
<td>Factor that indicates whether or not the structure is subjected to sidesway along the local Z axis of the member. The values are: 0 - Sidesway is present along the local Z-axis of the member 1 - There is no sidesway along the local Z-axis of the member. The sidesway condition is used to determine the value of $C_m$ explained in Section 4.1.1, page I-A-41 of the Aluminum specifications.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Member length</td>
<td>Spacing in the longitudinal direction of shear stiffeners for stiffened flat webs. It is input in the current units of length. See section 3.4.21 on page I-A-40 of the Aluminum specifications for information regarding this parameter.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| STRUCTURE      | 1             | In Table 3.4-1 in Section I-A of the Aluminum specifications, it is mentioned that the value of coefficients \( \nu, ny \) and \( na \) are dependent upon whether the structure being designed is a building or a bridge. Users may convey this information to STAAD using the parameter STRUCTURE. The values that can be assigned to this parameter are:  
1 - Buildings and similar type structures  
2 - Bridges and similar type structures |
| TRACK          | 2             | This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are:  
1 - Prints only the member number, section name, ratio, and PASS/FAIL status.  
2 - Prints the design summary in addition to that printed by TRACK 1  
3 - Prints the member properties and alloy properties in addition to that printed by TRACK 2.  
4 - Prints the values of variables used in design in addition to that printed by TRACK 3. |
| UNL            | Member length | Distance between points where the compression flange is braced against buckling or twisting. This value must be provided in the current units. This value is used to compute the allowable stress in bending compression. |
### D1.H.3.1 Aluminum Alloys available in STAAD

#### Table 110: Alloy Parameters

<table>
<thead>
<tr>
<th>Value</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1100-H12</td>
</tr>
<tr>
<td>2</td>
<td>1100-H14</td>
</tr>
<tr>
<td>3</td>
<td>2014-T6</td>
</tr>
<tr>
<td>4</td>
<td>2014-T6510</td>
</tr>
<tr>
<td>5</td>
<td>2014-T6511</td>
</tr>
<tr>
<td>6</td>
<td>2014-T651</td>
</tr>
<tr>
<td>7</td>
<td>3003-H12</td>
</tr>
<tr>
<td>8</td>
<td>3003-H14</td>
</tr>
<tr>
<td>9</td>
<td>3003-H16</td>
</tr>
<tr>
<td>10</td>
<td>3003-H18</td>
</tr>
<tr>
<td>11</td>
<td>3004-H32</td>
</tr>
<tr>
<td>12</td>
<td>3004-H34</td>
</tr>
<tr>
<td>13</td>
<td>3004-H36</td>
</tr>
</tbody>
</table>

In Table 3.4-2 in Section I-A of the Aluminum specifications, it is mentioned that the value of coefficients $K_t$ and $K_c$ are dependent upon whether or not, the location of the section where design is done is within 1.0 inch of a weld. The WELD parameter is used in STAAD for this purpose. The values that can be assigned to this parameter are:

- 0 - Region is farther than 1.0in from a weld
- 1 - Region is within 1.0in from a weld
<table>
<thead>
<tr>
<th>Value</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>3004-H38</td>
</tr>
<tr>
<td>15</td>
<td>5005-H12</td>
</tr>
<tr>
<td>16</td>
<td>5005-H14</td>
</tr>
<tr>
<td>17</td>
<td>5005-H32</td>
</tr>
<tr>
<td>18</td>
<td>5005-H34</td>
</tr>
<tr>
<td>19</td>
<td>5050-H32</td>
</tr>
<tr>
<td>20</td>
<td>5050-H34</td>
</tr>
<tr>
<td>21</td>
<td>5052-H32</td>
</tr>
<tr>
<td>22</td>
<td>5052-H34</td>
</tr>
<tr>
<td>23</td>
<td>5083-H111</td>
</tr>
<tr>
<td>24</td>
<td>5086-H111</td>
</tr>
<tr>
<td>25</td>
<td>5086-H116</td>
</tr>
<tr>
<td>26</td>
<td>5086-H32</td>
</tr>
<tr>
<td>27</td>
<td>5086-H34</td>
</tr>
<tr>
<td>28</td>
<td>5454-H111</td>
</tr>
<tr>
<td>29</td>
<td>5454-H112</td>
</tr>
<tr>
<td>30</td>
<td>5456-H111</td>
</tr>
<tr>
<td>31</td>
<td>5456-H112</td>
</tr>
<tr>
<td>32</td>
<td>6005-T5</td>
</tr>
<tr>
<td>33</td>
<td>6105-T5</td>
</tr>
<tr>
<td>34</td>
<td>6061-T6</td>
</tr>
<tr>
<td>35</td>
<td>6061-T6510</td>
</tr>
<tr>
<td>36</td>
<td>6061-T6511</td>
</tr>
<tr>
<td>37</td>
<td>6061-T651</td>
</tr>
<tr>
<td>38</td>
<td>6063-T5</td>
</tr>
<tr>
<td>39</td>
<td>6063-T6</td>
</tr>
</tbody>
</table>
# D1.H.4 Code Checking

The purpose of code checking is to determine whether the initially specified member properties are adequate to carry the forces transmitted to the member due to the loads on the structure. Code checking is done at the locations specified by either the \texttt{SECTION} command or the \texttt{BEAM} parameter described above.

See \texttt{T.1 – Steel Portal Frame} on page 373 for STAAD provides an example on the usage of the \texttt{CHECK CODE} command.

Refer to \texttt{D1.B.1.3 Code Checking} on page 1418 for general information on Code Checking. Refer to \texttt{TR.49 Code Checking Specification} on page 2852 for details the specification of the Code Checking command.

## D1.H.4.1 Example

Sample input data for Aluminum Design

```plaintext
PARAMETER
    CODE ALUMINUM
    BEAM 1 ALL
    KY 1.2 MEMB 3 4
    ALLOY 35 ALL
    PRODUCT 2 ALL
    TRACK 3 ALL
    SELECT ALL
PARAMETER
    CODE ALUMINUM
    ALCLAD 1 ALL
    STRUCT 1 ALL
    CHECK CODE ALL
```

# D1.H.5 Member Selection

The member selection process involves the determination of the least weight member that \texttt{PASSes} the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it.

See \texttt{T.1 – Steel Portal Frame} on page 373 for STAAD provides an example on the usage of the \texttt{SELECT MEMBER} command.

Refer to \texttt{D1.B.1.4 Member Selection} on page 1419 for general information on Member Selection. Refer to \texttt{TR.49.1 Member Selection Specification} on page 2853 for details the specification of the Member Selection command.

# D1.I. American Codes - Steel Design per ASCE Transmission Tower Codes

STAAD.Pro is capable of performing steel design based on the American Transmission Tower code ASCE 10-97 *Design of Latticed Steel Transmission Structures*.

**D1.I.1.1 General Comments**

The ASCE 10-97 code is meant to supercede the older edition of the code, available under the name ASCE Publication 52. However, in the interests of backward compatibility, both codes are currently accessible in STAAD.Pro.

Design is available for all standard sections listed in the AISC ASD 9th edition manual, namely, Wide Flanges, S, M, HP, Tees, Channels, Single Angles, Double Angles, Tubes and Pipes. Design of HSS sections (those listed in the 3rd edition AISC LRFD manual) and Composite beams (I shapes with concrete slab on top) is not supported.

To use the ASCE 52 code, use the commands

```
PARAMETER
CODE ASCE 52
```

To use the ASCE 10-97 code, use the commands

```
PARAMETER
CODE ASCE
```

The detailing requirements, such as provisioning of stiffeners and checking the local effects like flange buckling, web crippling, etc. must be performed manually. It is assumed that you are familiar with the basic concepts of Steel Design facilities available in STAAD. Please refer to **D. Steel Design** (on page 944) for detailed information on this topic.

**D1.I.1.2 Allowable Stresses per ASCE 10-97**

Member selection and code checking operations in the STAAD implementation of ASCE 10-97 are done to resist loads at stresses approaching yielding, buckling, fracture and other limiting conditions specified in the standard. Those stresses are referred to in the standard as Design Stresses. The appropriate sections of the ASCE standard where the procedure for calculating the design stresses is explained are as follows.

D1.I.1.2.1 Design Axial Tensile Stress

Design tensile stresses are calculated on the basis of the procedure described in section 3.10. The NSF parameter (see the Parameters table shown later in this section) may be used if the section area needs to be reduced to account for bolt holes.

D1.I.1.2.2 Design Axial Compressive Stress

Design compressive stress calculation is based on the procedures of section 3.6 through 3.9. For angle members under compression, the procedures of sections 3.7 and 3.8 have been implemented. Capacity of the section is computed for column buckling and wherever applicable, torsional buckling. The user may control the effective lengths for buckling using the LT, LY, LZ and/or KT, KY, KZ parameters (see the Parameters table shown later in this section).

D1.I.1.2.3 Design Bending Compressive Stress

Calculations for design bending compressive stress about the major axis and minor axis are based on the procedures of section 3.14. Procedures outlined in sections 3.14.1 through 3.14.6 have been implemented.

D1.I.1.2.4 Design Bending Tensile Stress
Calculations for design bending tensile stress about the major and minor axis are based on the procedures of section 3.14.2.

D1.1.1.2.5 Design Shear Stress
Calculation of the design shear stress is based on the procedure outlined in section 3.15 of the ASCE 10-97. The procedure of section 3.15.2 is followed for angles and the procedure of section 3.15.1 is followed for all other sections.

D1.1.1.3 Critical Conditions used as criteria to determine Pass/Fail status
These are Clause 3.4 for slenderness limits, Clause 3.12 for Axial Compression and Bending, Clause 3.13 for Axial Tension and Bending, Clause 3.9.2 for Maximum \( \frac{w}{t} \) ratios and Clause 3.15 for Shear.

D1.1.1.4 Design Parameters
Design parameters are summarized in the table shown later in this section. These parameters may be used to control the design process to suit specific modeling needs. The default parameter values have been selected such that they are frequently used numbers for conventional design.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 111: Steel Design Parameters for ASCE 10-97

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as ASCE to design per ASCE 10-97. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
</tbody>
</table>
| BEAM           | 1.0           | 0 = Perform design at beam ends and section locations specified according to the SECTION command  
1 = Perform design at the ends and eleven intermediate sections of the beam |
<p>| CMY CMZ        | 0.85 for sidesway and calculated for no sidesway | Cm value in local y and z axes as defined in equation 3.12-1 on p.10 of ASCE 10-97. |
| DMAX           | 4.50 in.      | Maximum allowable depth for member selection |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DBL</td>
<td>0.75 in.</td>
<td>Diameter of bolt for calculation of number of bolts required and the net section factor.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in.</td>
<td>Minimum allowable depth for member selection</td>
</tr>
<tr>
<td>ELA</td>
<td>4</td>
<td>Indicates what type of end conditions are to be used from among Equations 3.7-4 thru 3.7-7 to determine the KL/R ratio.</td>
</tr>
</tbody>
</table>
|                |               | 1. EQN.3.7-4, Page 4  
|                |               | 2. EQN.3.7-5, Page 4  
|                |               | 3. EQN.3.7-6, Page 4  
|                |               | 4. EQN.3.7-7, Page 5 |
|                |               | **Note:** Value 1 is valid for leg members only. |
| ELB            | 1             | Indicates what type of end conditions are to be used from among Equations. 3.7-8 thru 3.7-10 and 3.7-12 thru 3.7-14 to determine the KL/R ratio. |
|                |               | 1. EQN.3.7-8, Page 5, EQN. 3.7-12, Page 5  
|                |               | 2. EQN.3.7-9, Page 5, EQN. 3.7-13, Page 5  
<p>|                |               | 3. EQN.3.7-10, Page 5, EQN. 3.7-14,Page 5 |
| FVB            | 30 KSI        | Shear strength of bolt. |
| FYB            | 36 KSI        | Yield strength of bolt. |
| FYLD           | 36.0 KSI      | Yield Strength of steel |
| KT             | 1.0           | Effective length coefficient for warping restraint (clause 3.14.4, p. 11) |
| KY             | 1.0           | Effective length factor (K) for compression buckling about the Y-axis (minor axis) |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor (K) for compression buckling about the Z-axis (major axis)</td>
</tr>
<tr>
<td>LEG</td>
<td>0.0</td>
<td>This parameter is meant for plain angles.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. indicates that the angle is connected by both legs and allowable stress in axial tension is 1.0FYLD.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. indicates that the angle is connected only by the shorter leg and allowable tensile stress is computed per clause 3.10.2 as 0.9FYLD.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. indicates that the angle is connected by the longer leg.</td>
</tr>
<tr>
<td>LT</td>
<td>Member Length</td>
<td>Effective length for warping.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about the Y-axis (minor axis)</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about the Z-axis (major axis)</td>
</tr>
<tr>
<td>MAIN</td>
<td>2</td>
<td>Parameter that indicates the member type for the purpose of calculating the KL/R ratio (see clause 3.4, PAGE 3, ASCE 10-97)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Leg member, KL/R ≤ 150</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Compression member, KL/R ≤ 200</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. Tension member, KL/R ≤ 500</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4. Hanger member, KL/R ≤ 375 (Clause 3C.4, page 31)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5. Redundant member, KL/R ≤ 250</td>
</tr>
<tr>
<td></td>
<td></td>
<td>10. Do not perform the KL/R Check</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>NHL</td>
<td>0</td>
<td>Number of bolt holes on the cross section that should be used to determine the net section factor for tension capacity.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio that determines the cut off point for pass/fail status. A value below this quantity indicates PASS while a value greater than this quantity indicates FAILURE.</td>
</tr>
</tbody>
</table>
| SSY | 0.0 | 0.0 = Sidesway in local y-axis  
1.0 = No sidesway |
| SSZ | 0.0 | Same as above except in local z-axis |
| TRACK | 0.0 | 0.0 = Suppresses printing of allowable stresses  
1.0 = Prints all allowable stresses |
| UNB | Member Length | Unsupported length of the bottom flange for calculating flexural strength. Will be used only if flexural compression is on the bottom flange. |
| UNF | 1.0 | Same as UNL, but provided as a fraction of the member length |
| UNL | Member Length | Unsupported length of member for calculation of allowable bending stress |
| UNT | Member Length | Unsupported length of the top flange for calculating flexural strength. Will be used only if flexural compression is on the top flange. |

**Note:** All values must be provided in the current unit system.
D1.I.1.5 Code Checking and Member Selection

Both code checking and member selection options are available in the ASCE 10-97 implementation. In general, it may be noted that the concepts followed in MEMBER SELECTION and CODE CHECKING procedures are similar to that of the AISC based design.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.


STAAD.Pro is capable of performing steel design based on the ASCE Manuals and Reports on Engineering Practice No. 52 – Guide for Design of Steel Transmission Towers, Second Edition

D1.I.2.1 General Comments

The design philosophy and procedural logistics for member selection and code checking is based upon the principles of allowable stress design. Two major failure modes are recognized: failure by overstressing and failure by stability considerations.

The following sections describe the salient features regarding the process of calculation of the relevant allowable stresses and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economical section is selected based on the least weight criteria. The code checking part of the program also checks the slenderness requirements, the minimum metal thickness requirements, and the width-thickness requirements.

The detailing requirements, such as provisioning of stiffeners and checking the local effects like flange buckling, web crippling, etc. must be performed manually. It is assumed that you are familiar with the basic concepts of Steel Design facilities available in STAAD. Please refer to D. Steel Design (on page 944) for detailed information on this topic.

D1.I.2.2 Allowable Stresses per ASCE (Pub. 52)

The member design and code checking in the STAAD implementation of ASCE (Pub. 52) is based upon the allowable stress design method. Appropriate sections of this publication are referenced below.

D1.I.2.2.1 Allowable Axial Tensile Stress

Allowable tensile stresses are calculated on the basis of the procedure described in section 4.10. The NSF parameter (Refer to D1.I.2.3 Design Parameters (on page 1559)) may be used if the net section area needs to be used.

D1.I.2.2.2 Allowable Axial Compressive Stress

Allowable compressive stress calculation is based on the procedures of section 4.6 through 4.9. For angle members under compression, the procedures of sections 4.7 and 4.8 have been implemented. Capacity of the section is computed for column buckling and wherever applicable, torsional buckling. The user may control the effective lengths for buckling using the LX, LY, LZ and/or KX, KY, KZ parameters (Refer to D1.I.2.3 Design Parameters (on page 1559)).

D1.I.2.2.3 Allowable Bending Compressive Stress
Calculations for allowable bending compressive stress about the major axis and minor axis are based on the procedures of section 4.14. Procedures outlined in sections 4.14.1 through 4.14.6 have been implemented.

D11.2.2.4 Allowable Bending Tensile Stress

Calculations for allowable bending tensile stress about the major and minor axis are based on the procedures of Section 4.14.2.

D11.2.2.5 Allowable Shear Stress

Calculation of the allowable shear stress is based on the procedure outlined in section 4.15 of the ASCE Pub. 52. The procedure of section 4.15.2 is followed for angles and the procedure of section 4.15.1 is followed for all other sections.

D11.2.2.6 Critical Conditions used as criteria to determine Pass/Fail status

These are Clause 4.4 for slenderness limits, Equation 4.12-1 for Axial Compression and Bending, Equation 4.13-1 for Axial Tension and Bending, Clause 4.9.2 for Maximum w/t ratios and Clause 4.15 for Shear.

D11.2.3 Design Parameters

These parameters may be used to control the design process to suit specific modeling needs. The default parameter values have been selected such that they are frequently used numbers for conventional design.

Table 112: Steel Design Parameters for ASCE (Pub. 52) Based Design

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as ASCE 52. Design code to follow. See [Concrete Design-Parameter Specification](on page 2859).</td>
</tr>
<tr>
<td>BEAM</td>
<td>0.0</td>
<td>Specifies locations along member length at which member design is designed. 2.0 = use the section locations specified according to the SECTION command 3.0 = at the ends and eleven intermediate sections of the beam</td>
</tr>
<tr>
<td>DBL</td>
<td>0.75 in.</td>
<td>Diameter of bolt for calculation of number of bolts required and the net section factor.</td>
</tr>
<tr>
<td>DMAX</td>
<td>4.50 in.</td>
<td>Maximum allowable depth for member selection</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in.</td>
<td>Minimum allowable depth for member selection</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>ELA</td>
<td>4</td>
<td>Indicates what type of end conditions are to be used from among Equations 4.7-4 thru 4.7-7 to determine the KL/R ratio.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = EQN.4.7-4, Page 26 (Valid for leg members only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = EQN.4.7-5, Page 27</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 = EQN.4.7-6, Page 27</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4 = EQN.4.7-7, Page 27</td>
</tr>
<tr>
<td>ELB</td>
<td>1</td>
<td>Indicates what type of end conditions are to be used from among Equations. 4.7-8 thru 4.7-10 to determine the KL/R ratio.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = EQN.4.7-8, Page 27, EQN. 4.7-12, Page 28</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = EQN.4.7-9, Page 27, EQN. 4.7-13, Page 28</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 = EQN.4.7-10, Page 27, EQN. 4.7-14, Page 28</td>
</tr>
<tr>
<td>FYB</td>
<td>30 KSI</td>
<td>Shear strength of bolt.</td>
</tr>
<tr>
<td>FYB</td>
<td>36 KSI</td>
<td>Yield strength of bolt.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36.0 KSI</td>
<td>Yield Strength of steel</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td>Effective length coefficient for warping restraint (clause 4.14.4, pg 36)</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor (K) for compression buckling about the Y-axis (minor axis)</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor (K) for compression buckling about the Z-axis (major axis)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| **LEG**        | 0.0           | This parameter is meant for plain angles.  
|                |               | 3.0 = the angle is connected by both legs and allowable stress in axial tension is 1.0·FYLD  
|                |               | 4.0 = the angle is connected only by the shorter leg and allowable tensile stress is computed per Cl. 4.10.2 as 0.9·FYLD  
|                |               | 5.0 = the angle is connected by the longer leg |
| **LT**         | Member Length | Effective length for warping. |
| **LY**         | Member Length | Length to calculate slenderness ratio for buckling about the Y-axis (minor axis) |
| **LZ**         | Member Length | Length to calculate slenderness ratio for buckling about the Z-axis (major axis) |
| **MAIN**       | 2             | Parameter that indicates the member type for the purpose of calculating the KL/R ratio (See Cl. 4.4, p. 25)  
|                |               | 1 = Leg member (KL/r ≤ 150)  
|                |               | 2 = Compression member (KL/r ≤ 200)  
|                |               | 3 = Tension member (KL/r ≤ 500)  
|                |               | 4 = Hanger member per Cl. 4C.4, p. 43 (KL/r ≤ 375)  
|                |               | 5 = Redundant member (KL/r ≤ 250)  
|                |               | 10 = Do not perform the slenderness (KL/r) check |
| **NHL**        | 0             | Number of bolt holes on the cross section that should be used to determine the net section factor for tension capacity. |
| **NSF**        | 1.0           | Net section factor for tension members |
### D1.2.4 Code Checking and Member Selection

Both code checking and member selection options are available in the ASCE Pub. 52 implementation. In general, it may be noted that the concepts followed in MEMBER SELECTION and CODE CHECKING procedures are similar to that of the AISC based design.

Refer to [D1.B.1.3 Code Checking](#) (on page 1418) for general information on Code Checking. Refer to [TR.49 Code Checking Specification](#) (on page 2852) for details the specification of the Code Checking command.

Refer to [D1.B.1.4 Member Selection](#) (on page 1419) for general information on Member Selection. Refer to [TR.49.1 Member Selection Specification](#) (on page 2853) for details the specification of the Member Selection command.

### D1.J. American Codes - Steel Design per API 2A-WSD 2000


### D1.J.1 Design Operations

STAAD.Pro contains a broad set of facilities for the design of structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design;
- Specify whether to perform code checking or member selection;

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio that determines the cut off point for pass/fail status. A value below this quantity indicates PASS while a value greater than this quantity indicates FAILURE.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Level of detail in output</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0 = Suppresses printing of allowable stresses</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = Prints all allowable stresses</td>
</tr>
<tr>
<td>UNF</td>
<td>1.0</td>
<td>Same as UNL, but provided as a fraction of the member length</td>
</tr>
<tr>
<td>UNL</td>
<td>Member Length</td>
<td>Unsupported length of member for calculation of allowable bending stress</td>
</tr>
</tbody>
</table>
• Specify design parameter values, if different from the default values; and
• Specify design parameters to carry out joint checks.

These operations may be repeated any number of times depending upon the design requirements.

The basic process is as follows:

1. Define the STAAD model geometry, loading, and analysis.
2. Run the analysis and API design which creates the Geometry file (file extension .PUN) and give preliminary design results.
3. Check and modify the Geometry file as necessary.
4. Re-run the analysis to read the modified Geometry file for the final design results.

**D1.J.1.1 Limitations**

The parameter SELECT 1.0 should not be used while carrying out punching shear checks. It can be used in initial runs for member selection.

No classification of the joint is performed using the loading. For the initial run of an API code check, all joints will be assumed to be a T/Y joint. Refer to D1.J.8 Joint Design (on page 1572) for details.

No hydrostatic checks are performed.

**D1.J.1.2 Truss Members**

A truss member is capable of carrying only axial force. So in design, no time is wasted calculating the allowable bending or shear stresses, thus reducing design time considerably. Therefore, if there is any truss member in an analysis (like bracing or strut, etc.), it is wise to declare it as a truss member rather than as a regular frame member with both ends pinned.

**D1.J.2 Allowables per API Code**

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the American Petroleum Institute (API-RP2A) Code. The 21st edition of API Code, as published in 2007, is used as the basis of this design (except for tension stress).

**D1.J.2.1 Tension Stress**

Allowable tension stresses, as calculated in STAAD, are based on the API Code, clause (3.2.1-1).

Allowable tension stress on the net section

\[ F_t = 0.60 \cdot F_y \]

**D1.J.2.2 Shear Stress**

Beam Shear Stress

Allowable beam shear stress on the gross section must conform to Clause 3.2.4-2 of the API code:

\[ F_v = 0.4 \cdot F_y \]

The maximum applied beam shear stress is per Eqn 3.2.4-1:

\[ f_v = V / 0.5 \cdot A \]

Torsional Shear Stress
Allowable torsional shear stress per Eqn. 3.2.4-4:

$$F_{vt} = 0.4 \cdot F_y$$

$F_{vt}$ is the maximum torsional shear stress per Clause 3.2.4-3 of the API code.

**D1.J.2.3 Stress Due to Compression**

The allowable compressive stress on the gross section of axially loaded compression members is calculated based on the formula 3.2.2-1 in the API Code when the largest effective slenderness ratio, $K_l/r$ is less than or equal to $C_c$. If $K_l/r$ exceeds $C_c$, then the allowable compressive stress is increased as per formula (3.2.2-2) of the Code.

Where:

$$C_c = \frac{2\pi^2 E}{F_y}$$

For $D/t > 60$, the lesser of $F_{xe}$ or $F_{xc}$ is substituted for $F_{xy}$.

Where:

- $F_{xe}$ = the elastic local buckling stress calculated with $C$, the critical elastic buckling coefficient = 0.3 (3.2.2-3)
- $F_{xc}$ = the inelastic local buckling stress. (3.2.2-4)

**D1.J.2.4 Combined Compression and Bending**

Members subjected to both axial compression and bending stresses are proportioned to satisfy API formula 3.3.1-1 and 3.3.1-2 when $f_a/F_a > 0.15$, otherwise formula 3.3.1-3 applies. It should be noted that during code checking or member selection, if $f_a/F_a > 1.0$, the program does not compute the second 3.3.1-1/2.

**D1.J.2.5 Bending Stress**

The allowable bending stress for tension and compression for a symmetrical member loaded in the plane of its minor axis, as given in Clause 3.2.3 of the API code, is:

a. When $D/t \leq 1,500/F_y$ (Imperial Units),

$$F_b = 0.75F_y$$

b. When $1,500/F_y < D/t \leq 3,000/F_y$ (Imperial Units),

$$F_b = [0.84 - 1.74 F_y D/(Et)]F_y$$

c. When $3,000/F_y < D/t \leq 300$ (Imperial Units),

$$F_b = [0.72 - 0.58 F_y D/(Et)]F_y$$

**D1.J.2.6 Simple Joints: Capacity Checks**

A typical joint and the terms involved with the joint checks are given below:
Definitions

\[ \theta = \text{Brace included rage} \]
\[ g = \text{Gap between braces} \]
\[ t = \text{Brace wall thickness at intersection} \]
\[ T = \text{Chord wall thickness at intersection} \]
\[ d = \text{Brace outside diameter} \]
\[ D = \text{Chord outside diameter} \]
\[ \beta = \frac{d}{D} \]
\[ \gamma = \frac{D}{2T} \]
\[ \tau = \frac{t}{T} \]

Joint Validity

The validity range of the joints that are identified will be checked as per Cl. 4.3.1 of the code. The conditions to be checked for each joint are as given below:

\[ 0.2 \leq \beta \leq 1.0 \]
\[ 10 \leq \gamma \leq 50 \]
\[ 30^\circ \leq \theta \leq 90^\circ \]

\[ F_y = 90 \text{ ksi (500 MPa)} \]
\[ g/D > -0.6 \text{ (for K joints)} \]

If any of these conditions are not satisfied for the joint under consideration, the program issues a warning message corresponding to the invalid parameter(s). The program will, however, perform the joint checks as the code allows for the design of such joints with modified values of yield strength. You can use the FYLD parameter to reset the yield strength.

Joint Capacity
The capacity of the joint, both the axial capacity and the moment capacity is

The allowable capacity for brace axial load, \( P_a \), is evaluated as:

\[
P_a = Q_u Q_f \frac{F_{yc} T^2}{FSJ \sin \theta}
\]

The allowable capacity for brace bending moment, \( M_a \), is evaluated as:

\[
M_a = Q_u Q_f \frac{F_{yc} T^2 d}{FSJ \sin \theta}
\]

Where:

- \( F_y \) = the yield stress of the chord member at the joint (or 0.8 of the tensile stress, if less)
- \( FSJ \) = the factor of safety parameter (1.6 by default)
- \( Q_u \) and \( Q_f \) are the strength factor and the Chord factor that are to be determined based on the joint type. The strength factor, \( Q_u \), is to be determined as given in Section 4.3.3 of the code (ref. Table 4.3-1 of the API code).

\[
Q_f = \left[ 1 + C_1 \left( \frac{FSJ P_c}{P_y} \right) - C_2 \left( \frac{FSJ M_c}{M_p} \right) - C_3 A^2 \right]
\]

\[
A = \sqrt{\left( \frac{FSJ P_c}{P_y} \right)^2 + \left( \frac{FSJ M_c}{M_p} \right)^2}
\]

\( P_c \) = axial load

\( M_c = \sqrt{M_{ipb}^2 + M_{opb}^2} \)

\( C_1, C_2, \) and \( C_3 \) are factors determined by the following table:

<table>
<thead>
<tr>
<th>Joint Type</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( C_3 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>K joints under brace axial loading</td>
<td>0.2</td>
<td>0.2</td>
<td>0.3</td>
</tr>
<tr>
<td>T/Y joints under brace axial loading</td>
<td>0.3</td>
<td>0.0</td>
<td>0.8</td>
</tr>
<tr>
<td>X joints under brace axial loading</td>
<td>( \beta \leq 0.9 )</td>
<td>0.2</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>( \beta = 1.0 )</td>
<td>-0.2</td>
<td>0.0</td>
</tr>
<tr>
<td>All joints under brace moment loading</td>
<td>0.2</td>
<td>0.0</td>
<td>0.4</td>
</tr>
</tbody>
</table>

**Note:** For values of \( \beta \) between 0.9 and 1.0, coefficients are linearly interpolated between listed values.

For joints that are a mixture of K, X, or Y joints, the capacity of the joint is evaluated as a weighted average of the capacities of each joint.

In case the joint is subjected to combined axial load and bending moments (in-plane and/or out-of-plane), the program performs the following interaction check as given by Cl 4.3.6 of the code:

\[
\left| \frac{P}{P_a} \right| + \left( \frac{M}{M_a} \right)_{ipb}^2 + \left| \frac{M}{M_a} \right|_{opb} \leq 1.0
\]
D1.J.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in Table 22A.1. These parameters communicate design decisions from the engineer to the program.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the $K_Z$ value ($k$ value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the $K_Z$ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 113: American (API) Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as API Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
</tbody>
</table>
| BEAM           | 1.0           | Beam parameter:  
0.0 = design only for end moments or those at locations specified by the SECTION command.  
1.0 = calculate moments at twelfth points along the beam, and use the maximum $M_z$ location for design.  
2.0 = Same for BEAM 1.0, but additional check is made at each end. |
| CB             | 1.0           | $C_b$ value as used in Section 1.5 of AISC  
0.0 = $C_b$ value to be calculated  
Any other value will mean the value to be used in design |
<p>| CMY CMZ        | 0.85 for sidesway and calculated for no sidesway | Cm value in local y &amp; z axes |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DMAX</td>
<td>100.0 in</td>
<td>Maximum allowable depth</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0</td>
<td>Minimum allowable depth</td>
</tr>
<tr>
<td>FSJ</td>
<td>1.6</td>
<td>Factor of safety used for joint checks.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 ksi</td>
<td>Yield strength of steel.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 ksi</td>
<td>Yield strength of steel.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 ksi</td>
<td>Yield strength of steel.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 ksi</td>
<td>Yield strength of steel.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis. Typically the minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis. Typically the major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length in local Y-axis to calculate slenderness ratio.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Length in local Z-axis to calculate slenderness ratio.</td>
</tr>
</tbody>
</table>
| MAIN           | 0.0           | Design for slenderness.  
|                |               | 1.0 = Main member  
|                |               | 2.0 = Secondary member |
| NSF            | 1.0           | Net section factor for tension members. |
| RATIO          | 1.0           | Permissible ratio of the actual to allowable stresses |
| SSY            | 0.0           | Design for sidesway.  
|                |               | 0.0 = Sidesway in local y-axis  
|                |               | 1.0 = No sidesway |
| SSZ            | 0.0           | Design for sidesway in local z-axis |
| TRACK          | 0.0           | Controls the level of detail in the output:  
|                |               | 0.0 = Print design output at the minimum level of detail.  
|                |               | 1.0 = Print design output at an intermediate level of detail.  
|                |               | 2.0 = Print design output at the maximum level of detail. |
### Parameter Names, Default Values, and Descriptions

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNF</td>
<td>1.0</td>
<td>Same as above provided as a fraction of actual member length</td>
</tr>
<tr>
<td>UNL</td>
<td>Member Length</td>
<td>Unsupported length for calculating allowable bending stress</td>
</tr>
<tr>
<td>WELD</td>
<td>1</td>
<td>Weld type, as explained in section 3.1.1 of the API code.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = Welding is one side only except for wide flange or tee sections, where the web is always assumed to be welded on both sides.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.0 = Welding is both sides. For closed sections like pipe or tube, the welding will be only on one side.</td>
</tr>
<tr>
<td>WMIN</td>
<td>1.16 in.</td>
<td>Minimum thickness</td>
</tr>
<tr>
<td>WSTR</td>
<td>0.4 X FLYD</td>
<td>Allowable welding stress</td>
</tr>
</tbody>
</table>

**Note:** The parameters DMAX and DMIN are only used for member selection.

### D1.J.4 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate as per API. Code checking is done using the forces and moments at specific sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks, the critical condition of API code (like any of the API specifications for compression, tension, shear, etc.), the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value), the governing load case, and the location (distance from the start of the number of forces in the member) where the critical condition occurs.

Code checking can be done with any type of steel section listed in D1.A.3 Member Properties (on page 1367). Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

### D1.J.5 Member Selection

The program is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN which limits the maximum and minimum depth of the members.
• Member selection can be performed with all types of hollow steel sections.
• Selection of members whose properties are originally input from a user created table will be limited to sections in the user table.
• Member selection cannot be performed on members whose section properties are input as prismatic.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

**D1.J.6 Chord Selection and $Q_f$ Parameter**

$Q_f$ is a factor to account for the presence of nominal longitudinal stress in the chord. When calculating $Q_f$ for the joints, the moments used in the chord stress calculation will be from the computer node results and not the representative moments underneath the brace. If the moment varies significantly along the chord, it is more accurate to use the actual chord moment in the middle of the brace foot print. The tests reported in Reference I\(^1\) were performed with a constant moment along the chord. Thus for a local joint check, the local chord moment (under the brace) should be used.

STAAD calculates $Q_f$ based on the moment at the chord member. The chord member can be selected automatically by initial screening by the program (based on geometry and independent of loading) or specified in the External file.

In the automatic selection of the chord two collinear members (5 degree tolerance) are used to identify the chord. The chord is then selected from one of the two members based on the larger diameter then thickness or then by the minimum framing angle; for T joints the first member modeled will be selected as the chord.

You should confirm that the chord either be assigned by the program or the user is representative of the local chord moment for the brace in question.

**D1.J.6.1 Reference**


**D1.J.7 Tabulated Results of Steel Design**

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

- **Member**: the member number for which the design is performed.
- **TABLE**: AISC steel section name which has been checked against the steel code or has been selected.
- **RESULTS**: prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
- **CRITICAL COND**: the section of the AISC code which governs the design.
- **RATIO**: prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
- **LOADING**: provides the load case number which governed the design.
- **FX, MY, and MZ**: provide the axial force, moment in local Y-axis, and the moment in local Z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to
perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.

**LOCATION** specifies the actual distance from the start of the member to the section where design forces govern.

**Note:** If the parameter TRACK is set to 1.0, the program will block out part of the table and will print the allowable bending stressed in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV).

**D1.J.7.1 Example of Member Code Check output**

For TRACK 0.0 output:

```
STAAD.Pro CODE CHECKING - (API )
****************************
PROGRAM CODE REVISION V21_API_2000/1
ALL UNITS ARE - KN METE (UNLESS OTHERWISE NOTED)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
    FX       MY       MZ       LOCATION
=======================================================================
6 ST PIP40610.0 (BRITISH SECTIONS)  PASS  API 3.3.1-2  0.024  2
  2.76 T  0.00       5.12   3.00
7 ST PIP40610.0 (BRITISH SECTIONS)  PASS  API 3.3.1-3  0.078  2
  98.14 C  0.00       5.12   0.00
```

For TRACK 1.0 or TRACK 2.0 output:

```
14 ST PIP1938.0 (BRITISH SECTIONS)
  PASS  API 3.3.1-3  0.130  2
  67.16 C  0.00       0.29  4.24
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMB= 14, UNIT NEW-MMS ,L= 4243. AX= 4670. SZ= 208157. SY= 208157.</td>
</tr>
<tr>
<td>KL/R-Y= 64.6 CB= 1.00 YLD= 248.21 ALLOWABLE STRESSES: FCZ= 186.2</td>
</tr>
<tr>
<td>FTZ= 186.2 FTY= 186.2 FA= 117.6 FT= 148.9 FV= 99.3</td>
</tr>
</tbody>
</table>
```

**D1.J.7.2 Example of Joint Check output**

For TRACK 0.0 output:

```
STAAD.Pro - API JOINT CHECKS TO 21st edition.
-----------------------------
NODE NO: 7  CHORD NO: 7  BRACE NO: 10  RATIO: 0.049  PASS
NODE NO: 7  CHORD NO: 7  BRACE NO: 13  RATIO: 0.245  PASS
NODE NO: 7  CHORD NO: 11  BRACE NO: 14  RATIO: 0.222  PASS
```

For TRACK 2.0 output:

```
STAAD.Pro - API JOINT CHECKS TO 21st edition.
-----------------------------
```
D1.J.8 Joint Design

**D1.J.8.1 Joint Checking**

The design of joints is based on Section 4 of the API code.

The program only checks simple joints and overlapping joints formed between circular hollow section members. Any other type of joint within the structure or joint cans will not be considered for API joint checks. Other types of joints (such as grouted joints, joints with ring stiffeners, etc.) are not considered.

**Material Strength**

The API code states in Cl. 4.2.1 that the value of yield stress of the chord member to be used in the calculation of the joint capacity should be limited to 0.8 times the tensile strength of the chord for materials with a yield stress less than or equal to 500 MPa.

The yield stress to be used in the joint capacity checks value is specified in the joint data file (*filename*.PUN). For every joint, the value specified in the FYLD column will be used as the yield strength to be used for the joint capacity checks. When the file is created for the first time by the program, a default value of 36 ksi is used for all joints. The value used for each joint check will also be reported in the output file.

**Note:** All the fields in the joint data file (*.PUN file) are to be in imperial units.

**Minimum Joint Capacity**

Clause 4.2.3 of the code specifies a minimum capacity for any joint as follows:

The connections at the ends of a member should develop the strength required by the design loads, but should not be less than 50% of the effective strength of the member. The effective strength is defined as the buckling
load for a compression member or the yield load for members in tension. You, however, must ensure that this condition is satisfied even if the joint strength indicates a PASS status.

The program checks to see if the capacity of a joint as calculated by the methods in the code satisfies this requirement. If not the program issues a warning to that effect and marks the joint as FAILED. The program calculates the axial and/or bending moment capacities of the joint and reports the load/capacity ratio for each condition. The program also reports a “critical ratio” along with the condition that induces this ratio. Note that the maximum among the various individual ratios will be reported as the ‘critical ratio’. The program also reports a PASS/FAIL status for the joint.

Refer to D1.J.2.6 Simple Joints: Capacity Checks (on page 1564) for details of capacity checks performed.

Joint Classification

Clause of 4.2.4 of the API code essentially classifies a joint into one of the three basic types: K, X, and Y. Joint classification is the process whereby the axial load in a given brace is subdivided into its K, X, and Y components corresponding to the three joint types. A joint—as considered in the code—is the connection between a "chord" and a "brace" that are in the same plane. The program considers any two members to be in the same plane if they lie in planes that are within ±15 degrees of each other. The classification of a joint can also be a mixture of any of the basic types mentioned above. Once the classification of a joint has been identified, the capacity of that joint is then evaluated per Section 4.3 of the code.

The program automatically identifies the joints in a structure and identifies the chord and the brace members. The program applies the ±15° rule to determine the members in a plane and then determines the joint as being the intersection point of these members. Since a joint is between a chord and a brace member, the program considers two members at a time and then proceeds to identify the chord and the brace member at that joint. The program assumes the member with the larger diameter among the two members as the chord member and the other is considered as the brace. If both members have the same diameter, the chord is assumed to be the member with the thicker wall. If both the diameter and thickness of the members are identical, the program will assume the most horizontal member to be the chord. To be automatically considered as a chord member, the member has to be continuous across the joint. The user can always edit the joint data file (*.PUN) to add or delete new BRACE-CHORD joints.

The chord and brace member numbers (from the STAAD input file) are saved under the CHORD and BRACE columns in the filename.PUN file.

When the joint data file (.PUN) is created by the program, a default joint Class Y is assumed for the initial joint checks. This is indicated by the K, X, and Y column values being set to 0, 0, and 1 respectively. Since the API code allows for a mixed joint classification, you must manually vary the contribution factors for K, X, and Y joint classes for a given joint. For example, if a joint is to be 25% K, 25% X, and 50% Y, then you must assign K column value of 0.25, X column value of 0.25, and a Y column value of 0.50 for that joint. The program will verify that the supplied contributions sum to 1.0.

If the joint has a gap (i.e., a K-GAP joint), the gap distance (in inches) must be supplied in the GAP column. The value to be provided will be the actual gap between the brace members at the joint. An overlap can be specified by setting the gap to a negative value. The overlapping brace in this case can then be indicated by specifying the member number at the OBRACE (Overlapping brace) column in the data file.

Overlapping Joints

Clause 4.4 of the API discusses overlapping joints. Checks for overlapping joints will be performed as described Section 22A.2.6 (on page 1564). The difference will be in that the gap value, g, will be taken as negative in evaluating the various factors.

If the axial loads in the overlapping brace and the through brace have the same sign, the axial load in the through brace will be increased to allow for the loads in the overlapping brace. This will be achieved by allowing a
portion of the overlapping brace load equal to the proportion of the overlapping brace area to be added to the axial load in the through brace.

**Note:** The program issues a warning for any joint overlap is less than $0.25 \beta D$.

### D1.J.8.2 Joint File Format

The data contained in the `filename`.PUN file should meet the following format. The overall process of performing punching shear checks consists of two steps which are explained in D1.J.8.1 Joint Checking (on page 1572).

When the API design module is invoked, the program will initially check for the presence of a `filename`.PUN file (where `filename` is the name of the .std file) in the same folder as the input file. If the program does not find such a file, it assumes that the joint design is being run for the first time and will create this file. If the program does find this file, it will assume that the joint design has been run at least once and will attempt to read the input data from this file. Not that modifying and saving the main structure (i.e., any changes to the main model using GUI or text editor) will invalidate all design results and the program will automatically delete all design related files including the *.PUN file. Hence if the user wishes to keep an existing version of the *.PUN file, he/she must make a separate copy of this file before making any changes to the model.

**Note:** Units used in this file must be kips and inches.

#### General Format

```
*BRACE CHORD K X Y D T d t GAP FYLD OBRACE TW SWAP
b# c# K% X% Y% Dc Tc db tb gap fy ob tw swap
```

Where:

- `b#` = the brace member number
- `c#` = the chord member number
- `K%`, `X%`, and `Y%` = The fractional contributions of K-type, X type and Y-type, respectively. Initially the joints will be classed as Y (i.e., K=0, X=0 and Y=1).
- `db`, `tb` = Diameter and thickness of BRACE member
- `Dc`, `Tc` = Diameter and thickness of CHORD member
- `gap` = Distance required to calculate gap factor for K bracing. Initially, the value of GAP is assumed as 0. An overlap can be specified by setting the gap to a negative value.
- `fy` = the yield stress to be used in the joint capacity checks
- `ob` = member number of the overlapping brace in an overlap joint (i.e., a gap value less than zero)
- `tw` = Used in overlap K-joint, taken as the lesser of the weld throat thickness or thickness `t` of the thinner brace in inches
- `swap` = If parameter SWAP 0 is used then major moment `Mz` is taken for In Plane Bending (IPB). SWAP 1 uses the minor moment `My` as the IPB.

#### Example

```
*BRACE CHORD K X Y D T d t GAP FYLD OBRACE TW SWAP
 10  7  0.000  0.000  1.000  16.000  0.394  16.000  0.394   0.00  36.0  0.0  0.00 0
 13  7  0.000  0.000  1.000  16.000  0.394   7.626  0.315   0.00  36.0  0.0  0.00 0
 14 11  0.000  0.000  1.000  16.000  0.394   7.626  0.315   0.00  36.0  0.0  0.00 0
```
D1.K. American Codes - Steel Design per ANSI/AISC N690 Design Codes


Design of members per ANSI/AISC N690-1994 requires the STAAD Nuclear Design Codes SELECT Code Pack.

D1.K.1.1 General Comments

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by ANSI/AISC N690-1994 and as amended by ANSI/AISC N690 1994(R2004)s2.

All the design steps are done as described in section 2.3 Allowable per AISC-ASD (Ninth Edition) Code of Technical Reference manual except for allowable stress in compression for AUSTENITIC STAINLESS STEEL. Section Q1.5.9 is used to calculate allowable compressive stress for Austenitic Stainless Steel. Correction made in Supplementary s1 published in April 15, 2002 has been applied.

Note: By default, N690 code uses Stainless Steel material in the design. Care should be taken to assign the proper Stainless Steel material properties to the members for the analysis. There is a parameter – STYPE – to change material type to either Stainless Steel (STYPE=1) or Carbon Steel (STYPE=0).

D1.K.1.1.1 Design Process

Members subjected to both axial compression and bending stresses are proportioned to satisfy equation Q1.6-1a:

\[
\frac{f_a}{SFC \cdot F_a} + \frac{f_{by}}{SMY \cdot F_{by}} + \frac{C_{mz}f_{bz}}{SMZ \cdot F_{bz} \left(1 - \frac{f_a}{F_{ez}}\right)} \leq 1.0
\]

and Q1.6-1b:

\[
\frac{f_a}{SFC \cdot 0.6F_y} + \frac{f_{by}}{SMY \cdot F_{by}} + \frac{f_{bz}}{SMZ \cdot F_{bz}} \leq 1.0
\]

when, \(f_a/F_a > 0.15\), as per section Q1.6.1 of the code.

Otherwise, equation Q1.6-2 must be satisfied:

\[
\frac{f_a}{SFC \cdot F_a} + \frac{f_{by}}{SMY \cdot F_{by}} + \frac{f_{bz}}{SMZ \cdot F_{bz}} \leq 1.0
\]

It should be noted that during code checking or member selection, if \(f_a/F_a\) exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for side-sway and \([0.6 \cdot 0.4 \cdot (M1/M2)]\), but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy equation Q1.6-3:

\[
\frac{f_a}{SFT \cdot 0.6F_y} + \frac{f_{by}}{SMY \cdot F_{by}} + \frac{f_{bz}}{SMZ \cdot F_{bz}} \leq 1.0
\]
Where:

SFC, SFT, SMZ, and SMY are stress limit coefficient parameters used to control the components of the interaction equations. Refer to D1.K.1.2 Design Parameters (on page 1576) for details.

D1.K.1.2 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure.

Table 114: Design Parameters for ANSI/AISC N690-1994

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as AISC N690 Design code to follow. See TR48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1</td>
<td>Beam parameter 0. Perform design at ends and those locations in the SECTION command. 1. Perform design at ends and at 1/12th section locations along the member length.</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>Used for Deflection Check only (i.e., when DFF is specified). 0. Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1. Deflection check based on the principle that maximum deflection is of the cantilever type</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Bending coefficient dependent upon moment gradient, as specified in Chapter F of AISC ASD. 0.0 = CB is calculated itself. Any other user-defined value is accepted.</td>
</tr>
<tr>
<td>CMY, CMZ</td>
<td>0.85</td>
<td>Cm value in local y &amp; z axes for sidesway and calculated for no sidesway</td>
</tr>
<tr>
<td>COMPOSITE</td>
<td>0</td>
<td>Composite action with connectors (CMP) 0. No composite action 1. Composite action 2. Ignore positive moments during design</td>
</tr>
<tr>
<td>CONDIA</td>
<td>0.625 in</td>
<td>Diameter of shear connectors (DIA), in current units.</td>
</tr>
<tr>
<td>CONHEIGHT</td>
<td>2.5 in</td>
<td>Height of shear connectors after welding (HGT), in current units.</td>
</tr>
<tr>
<td>CYCLES</td>
<td>500,000</td>
<td>Cycles of maximum stress to which the shear connector is subject (CYC).</td>
</tr>
<tr>
<td>DFF</td>
<td>None(Mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DLR2</td>
<td>0.4</td>
<td>Ratio of moment due to dead load applied after the concrete hardens to the total moment (DR2).</td>
</tr>
<tr>
<td>DLRATIO</td>
<td>0.4</td>
<td>Ratio of moment due to dead load applied before the concrete hardens to the total moment (DR1).</td>
</tr>
<tr>
<td>DMAX</td>
<td>4.5 inch</td>
<td>Maximum allowable depth</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 inch</td>
<td>Minimum allowable depth</td>
</tr>
<tr>
<td>EFFWIDTH</td>
<td>1/4 Member Length</td>
<td>Effective width of concrete slab (WID).</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 KSI</td>
<td>Yield strength of steel in current units.</td>
</tr>
<tr>
<td>FPC</td>
<td>3 KSI</td>
<td>Compressive strength of concrete at 28 days, in current units.</td>
</tr>
</tbody>
</table>
| FSS            | 1             | Full section shear for welding.  
|                |               | 0. False  
|                |               | 1. True |
| FU             | 60 KSI        | Ultimate tensile strength of steel, in current units. |
| FYLD           | 46 KSI        | Yield strength of steel, in current units. |
| KX             | 1.0           | Effective length factor for flexural torsional buckling. |
| KY             | 1.0           | Effective Length Factor for Compression in local y-axis. Usually, this is minor axis. |
| KZ             | 1.0           | Effective Length Factor for Compression in local z-axis. Usually, this is major axis. |
| LX             | Member Length | Length for flexural torsional buckling. |
| LY             | Member Length | Length to calculate slenderness ratio (KL/r) for buckling about local Y axis. |
| LZ             | Member Length | Same as above except in z-axis (major). |
| MAIN           | 0.0           | Design for slenderness:  
|                |               | 0. check for slenderness  
<p>|                |               | 1. suppress slenderness check |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section Factor for tension members</td>
</tr>
<tr>
<td>OVR</td>
<td>1.0</td>
<td>Factor by which all allowable stresses/capacities should be multiplied. Default of 1.0 indicates that no overstressing is allowed.</td>
</tr>
<tr>
<td>PLTHICK</td>
<td>0</td>
<td>Thickness of the cover plate welded to the bottom flange of the composite beam (PLT), in current units.</td>
</tr>
<tr>
<td>PLTWIDTH</td>
<td>0</td>
<td>Width of the cover plate welded to the bottom flange of the composite beam (PLT), in current units.</td>
</tr>
<tr>
<td>PROFILE</td>
<td>None</td>
<td>Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
</tr>
<tr>
<td>RIBHEIGHT</td>
<td>0</td>
<td>Height of ribs of form steel deck (RBH), in current units.</td>
</tr>
<tr>
<td>RIBWIDTH</td>
<td>0</td>
<td>Width of ribs of form steel deck (RBW), in current units.</td>
</tr>
<tr>
<td>SFC</td>
<td>1.0</td>
<td>Stress limit coefficient for compression (SLC) as found in Table Q 1.5.7.1.</td>
</tr>
<tr>
<td>SFT</td>
<td>1.0</td>
<td>Stress limit coefficient for tension (SLC) as found in Table Q 1.5.7.1.</td>
</tr>
</tbody>
</table>
| SHE            | 0             | Shear stress calculation option  
0. Computes the actual shear stress using VQ/It  
1. Computes the actual shear stress using V(Ay or Az) |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| SHORING        | 0             | Temporary shoring during construction  
|                |               | 0. Without shoring  
|                |               | 1. With shoring  |
| SLABTHICK      | 4 in          | Thickness of concrete slab or thickness of concrete slab above the form steel deck (THK), in current units. |
| SMY            | 1.0           | Stress limit coefficient for minor axis bending (SLC) as found in Table Q 1.5.7.1. |
| SMZ            | 1.0           | Stress limit coefficient for major axis bending (SLC) as found in Table Q 1.5.7.1. |
| SSY            | 0             | Design for sidesway in the local y axis.  
|                |               | 0. Sidesway  
|                |               | 1. No sidesway  |
| SSZ            | 0             | Design for sidesway in the local z axis.  
|                |               | 0. Sidesway  
|                |               | 1. No sidesway  |
| STIFF          | Member length or depth whichever is greater | Spacing of stiffeners for plate girder design, in current units. |
| STYPE          | 0.0           | Type of steel material  
|                |               | 0. Normal Steel  
|                |               | 1. Austenitic Stainless Steel  |
| TAPER          | 1             | Design for tapered member.  
|                |               | 0. Design for tapered I-section based on rules in Chapter F and Appendix B.  
<p>|                |               | 1. Design for tapered section based on Appendix F.  |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| TMAIN          | 240 for main member  
                 300 for “Truss” member | Slenderness limit under tension |
| TORSION        | 0             | Design for torsion.  
                 0. Do not design for torsion.  
                 1. Design for torsion. |
| TRACK          | 0.0           | Controls the levels of detail to which results are reported.  
                 0. Minimum detail  
                 1. Intermediate detail level  
                 2. Maximum detail |
| UNB            | Member Length | Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange. |
| UNT            | Member Length | Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange. |
| WELD           | 1             | Design for weld.  
                 0. Closed sections.  
                 1. Open sections. |
| WMAX           | 1 in          | Maximum weld thickness, in current units. |
| WMIN           | 0.625 in      | Minimum weld thickness, in current units. |
| WSTR           | 0.4·Fyld      | Allowable welding stress, in current units. |

D1.K.1.2.1 Notes

1. All values are entered in the current units
2. parameters DMAX and DMIN are only used with the MEMBER SELECTION command

For code checking of steel members, STAAD compares the actual stresses with the allowable stresses as defined by ANSI/AISC N690-1984: *Nuclear Facilities - Steel Safety-Related Structures for Design, Fabrication, and Erection.* A brief description of some of the major allowable stresses is described herein.

**D1.K.2.1 Design Process**

The following Checks are to be performed on a Steel Member as per this AISC N690 – 1984 Code. When a design is performed, the output file the reports the maximum utilization from all of the checks.

**D1.K.2.1.1 Slenderness**

The maximum allowable slenderness ratio in Compression $(K\cdot L/r_{min})$, as per clause Q1.8.4 of the code shall not exceed 200. And the maximum allowable slenderness ratio in Tension $(L/r_{min})$ shall not exceed 240 for main members and 300 for bracing members and other secondary members.

This can be controlled by using the existing MAIN and TMAIN parameters respectively.

The default value of MAIN is 200 and for TMAIN is 240.

**D1.K.2.1.2 Check for Element Slenderness and Stress Reduction Factors**

The permissible Width-to-Thickness Ratio of “Un-stiffened Elements under Compression” is determined as per section Q1.9.1 and that of “Stiffened Elements under Compression” is determined as per section Q1.9.2 of the code.

The permissible Width–Thickness Ratio of web is determined as per section Q1.10.2.

**D1.K.2.1.3 Tension**

Allowable tensile stress on the Net section is calculated as $0.60\cdot F_y$, but not more than $0.5\cdot F_u$ on the Effective Net area, as per section Q1.5.1.1.

The Net Area $(A_n)$ shall be determined in accordance with Q1.14, and the NSF parameter can be utilized for that.

The Effective Net Area $(A_e)$ of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. Q1.14),

$$A_e = C_t \cdot A_n$$

Unless otherwise specified, the default value of the CT parameter is set as 0.75.

The value of CT parameter for other conditions is described at section Q1.14.

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

**D1.K.2.1.4 Compression**

The allowable compressive stress for columns which meet the provisions of section Q1.9, except those fabricated from austenitic stainless steel shall be as required by Q1.5.1.3. The allowable compressive stress for columns fabricated from austenitic stainless steel shall be in accordance to section Q1.5.9.

**A. Gross Sections of Columns, except those fabricated of austenitic stainless steel:**

**a.** On gross section of axially loaded compression members, when $(Kl/r) \leq C_c$,

$$F_a = \frac{[1 - (Kl/r)^2/(2\cdot C_c^2)]F_y}{\{5/3 + [3(Kl/r)/(8\cdot C_c)] - [(Kl/r)^3/(8\cdot C_c^3)]\}}$$

Where:

$$C_c = \left(\frac{2\pi^2E}{F_y}\right)^{1/2}$$
b. When \((Kl/r) > C_c\),
\[F_a = 12 \pi^2 E /[23(\text{KL/r})^2]\]

B. Gross sections of columns fabricated from Austenitic Stainless steel:

a. When \((Kl/r) \leq 120\),
\[F_a = F_y/2.15 - (F_y/2.16 - 6)/120(kL/r)\]
b. When \((Kl/r) > 120\),
\[F_a = 12 - (KL/r)/20\]

If the provisions of the section Q1.9 are not satisfied,

A. For un-stiffened compression element, a reduction factor \(Q_s\) is introduced. Detailed values of \(Q_s\) for different shapes are given in Section QC2.

B. For stiffened compression element, a reduced effective width \(b_e\) is introduced.

a. For the flanges of square and rectangular sections of uniform thickness:
\[b_e = 253 \cdot t/\sqrt{F_y} \cdot \{1 - (50.3/[t/b\sqrt{F_y}]) \leq b\}\]
b. For other uniformly compressed elements:
\[b_e = 253 \cdot t/\sqrt{F_y} \cdot \{1 - (44.3/[t/b\sqrt{F_y}]) \leq b\}\]

Consequently, a reduction factor \(Q_a\) is introduced and is equal to the effective area divided by the actual area. Combining both these factors, allowable stress for axially loaded compression members containing stiffened or unstiffened elements shall not exceed
\[F_a = Q_s Q_a [1 - (KL/r)^2/(2C_c^2)] F_y / \{5/3 + [3(KL/r)/(8C_c)] \cdot [(KL/r)^3/(8C_c^3)]\}\]

Where:
\[C_c' = [(2 \pi^2 E)/(Q_s Q_a F_y)]^{1/2}\]

D1.K.2.1.5 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in section Q1.5.1.4 is:

A. Along Major Axis:

a. Tension and compression on extreme fibers of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection Q1.5.1.4.1.1 to 7, shall result in a maximum bending stress:
\[F_b = 0.66 \cdot F_y\]

If meeting the requirements of this member of:

a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed 65/\sqrt{F_y}.

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed 190/\sqrt{F_y}.

c. The depth-thickness ratio of the web shall not exceed
\[d/t = (640/\sqrt{F_y})[1 - 3.74(f_a/F_y)] \text{ when } f_a/F_y \leq 0.16\]
\[d/t = 257/\sqrt{F_y} \text{ when } f_a/F_y > 0.16\]

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of 76b_f/\sqrt{F_y} nor 20000/(d/A_f)F_y.

b. For noncompact and slender elements, section Q1.5.1.4.2 is followed.
c. For box-type flexural members, maximum bending stress is:

\[ F_b = 0.60 \cdot F_y \]

B. Along Minor Axis:

a. For doubly symmetrical members (I shaped) meeting the requirements of section Q1.5.1.4.1, maximum tensile and compressive bending stress shall not exceed the following value as per section Q1.5.1.4.3:

\[ F_b = 0.75 \cdot F_y \]

b. For doubly symmetrical members (I shaped) meeting the requirements of section Q1.5.1.4.1, except where \( b_f/2t_f > 65/\sqrt{F_y} \) but is less than \( 95/\sqrt{F_y} \), maximum tensile and compressive bending stress shall not exceed:

\[ F_b = F_y [0.79 - 0.002(b_f/2t_f)\sqrt{F_y}] \]

D1.K.2.1.6 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy equation Q1.6-1a:

\[
\frac{f_a}{SFC \cdot F_a} + \frac{C_{my}f_{by}}{SMY \cdot F_{by} \left(1 - \frac{f_a}{F_y}\right)} + \frac{C_{mz}f_{bz}}{SMZ \cdot F_{bz} \left(1 - \frac{f_a}{F_y}\right)} \leq 1.0
\]

and Q1.6-1b

\[
\frac{f_a}{SFC \cdot 0.6F_y} + \frac{f_{by}}{SMY \cdot F_{by}} + \frac{f_{bz}}{SMZ \cdot F_{bz}} \leq 1.0
\]

when, \( f_a/F_a > 0.15 \), as per section Q1.6.1 of the code.

Otherwise, equation Q1.6-2 must be satisfied:

\[
\frac{f_a}{SFC \cdot F_a} + \frac{f_{by}}{SMY \cdot F_{by}} + \frac{f_{bz}}{SMZ \cdot F_{bz}} \leq 1.0
\]

It should be noted that during code checking or member selection, if \( f_a/F_a \) exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient \( C_m \) is taken as 0.85 for side-sway and \( [0.6 - 0.4(M1/M2)] \), but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy equation Q 1.6-1b:

\[
\frac{f_a}{SFT \cdot 0.6F_y} + \frac{f_{by}}{SMY \cdot F_{by}} + \frac{f_{bz}}{SMZ \cdot F_{bz}} \leq 1.0
\]

Where SFC, SFT, SMZ, and SMY are stress limit coefficient parameters used to control the components of the interaction equations. Refer to D1.K.2.3 Design Parameters (on page 1585) for details.

D1.K.2.1.7 Shear Stress

Allowable shear stress on the gross section [ref. section Q1.10.5.2] is calculated as

\[ F_v = (F_y/2.89)C_v \leq 0.4 \cdot F_y \]

Where:

\[ C_v = (45,000 \cdot k)/[F_y(h/t)^2], \text{ when } h/t \leq 0.8 \]

\[ C_v = (190/(h/t)^{\frac{3}{2}})/(k/F_y), \text{ when } h/t > 0.8 \]
\( k = 4.00 + \frac{5.34}{(a/h)^2} \), when \( a/h \leq 1.0 \)

\( k = 5.34 + \frac{4.00}{(a/h)^2} \), when \( a/h > 1.0 \)

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

**D1.K.2.2 Member Property Specification**

For specification of member properties, the specified steel section available in Steel Section Library of STAAD may be used, namely: I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, and Double Channel sections.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to [G.6 Member Properties](#) (on page 2322).

**D1.K.2.3 Design Parameters**

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure.

**Table 115: Design Parameters for ANSI/AISC N690-1984**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as AISC N690 1984 to use the ANSI/AISC N690-1984 code for checking purposes. Design code to follow. See <a href="#">TR.48.1 Parameter Specifications</a> (on page 2851).</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Bending coefficient dependent upon moment gradient. 0.0 = CB is calculated itself. Any other user-defined value is accepted.</td>
</tr>
<tr>
<td>CMY, CMZ</td>
<td>0.85</td>
<td>Cm value in local y &amp; z axes.</td>
</tr>
<tr>
<td>CT</td>
<td>0.75</td>
<td>Reduction Coefficient in computing net effective net area of an axially loaded tension member.</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maximum allowable local deflection.</td>
</tr>
<tr>
<td>DJ1, DJ2</td>
<td>Start Joint of member, End Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot;.</td>
</tr>
<tr>
<td>DMAX</td>
<td>45 inch</td>
<td>Maximum allowable depth.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 inch</td>
<td>Minimum allowable depth.</td>
</tr>
<tr>
<td>FU</td>
<td>60 KSI</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 KSI</td>
<td>Yield strength of steel in current units.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective Length Factor for Compression in local y-axis. Usually, this is minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective Length Factor for Compression in local z-axis. Usually, this is major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about local Y axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in z-axis (major).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| MAIN           | 0.0           | Design for slenderness.  
|                |               | 0. Check for slenderness  
|                |               | 1. Suppress slenderness check |
| NSF            | 1.0           | Net section Factor for tension members |
| PROFILE        | None          | Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2851) for details. |
| RATIO          | 1.0           | Permissible ratio of the actual to allowable stresses. |
| SFC            | 1.0           | Stress limit coefficient for compression (SLC) as found in Table Q 1.5.7.1. |
| SFT            | 1.0           | Stress limit coefficient for tension (SLC) as found in Table Q 1.5.7.1. |
| SMY            | 1.0           | Stress limit coefficient for minor axis bending (SLC) as found in Table Q 1.5.7.1. |
| SMZ            | 1.0           | Stress limit coefficient for major axis bending (SLC) as found in Table Q 1.5.7.1. |
| STIFF          | Member length or depth whichever is greater | Spacing of stiffeners for plate girder design |
| STYPE          | 0.0           | Steel type:  
|                |               | 0.0 = Normal Steel  
|                |               | 1.0 = Austenitic Stainless Steel |
| TMAIN          | 240 for main member  
|                | 300 for “Truss” member | Slenderness limit under tension |
### Parameter Name | Default Value | Description
--- | --- | ---
TRACK | 0.0 | Controls the levels of detail to which results are reported.  
0 = Minimum detail  
1 = Intermediate detail level  
2 = Maximum detail

UNB | Member Length | Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.

UNT | Member Length | Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.

### D1.K.2.3.1 Notes
1. All values are entered in the current units
2. Parameters DMAX and DMIN are only used with the MEMBER SELECTION command

### D1.K.2.4 Code Checking and Member Selection
Both code checking and member selection options are available with the AISC N690 1984 code.
Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.
Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

### D1.L. American Codes - Steel Design per ASME NF Codes
The following American Society of Mechanical Engineers – Nuclear Facility codes are available.

#### D1.L.1. ASME NF 3000 - 1974 & 1977 Codes
STAAD.Pro is capable of performing steel design based on the American Society of Mechanical Engineers Nuclear Facility Code, ASME NF 3000 - 1974 & 1977.

**Note:** From design point of view, there are no major differences between NF-3000 1974 and NF-3000 1977 version of codes.

**D1.L.1.1 Design Process**

The design process follows the following design checks:

Each one of the checks are described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

**D1.L.1.1.1 Slenderness**

As per clause XVII-2223 of NF-3000 1974, the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

**D1.L.1.1.2 Tension**

Allowable tensile stress on the Net section is calculated as \(0.60 \times F_y\), but not more than \(0.5\times F_u\) on the Net area.

The Net Area \(A_n\) shall be determined in accordance with the clause XVII-2283 of NF-3000 1974, and the NSF parameter can be utilized for that.

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.Pro.

**D1.L.1.1.3 Compression**

The allowable compressive stress for columns shall be as required by clause XVII-2213 of NF-3000 1974.

**a. Gross Sections of Columns:**

- **On gross section of axially loaded compression members, when \((KL/r) < C_c\),**

\[
F_a = F_y \frac{1 - \left(\frac{KL}{r}\right)^2}{2C_c^2} \left[ 3 + \frac{3KL}{8C_c} \cdot \left(\frac{KL}{r}\right)^3 \right]
\]

Where:

\[
C_c = \sqrt{\frac{2\pi^2 E}{F_y}}
\]

- **When \((KL/r) > C_c\),**

\[
F_a = \frac{12\pi^2 E}{23(KL/r)^2}
\]

- **When \((KL/r) > 120\),**

\[
F_{a_{0}} = \frac{F_{a_{0}}[Eq.(a1) or Eq.(a2)]}{1.6 \cdot \left(\frac{l}{200r}\right)}
\]

**b. Member elements other than columns:**

- **For Plate Girder Stiffeners**, \(F_a = 0.60 \cdot F_y\)
- **For webs of rolled shapes**, \(F_a = 0.75 \cdot F_y\)

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the sub-sections of clause XVII-2224 of NF-3000 1974.

If the above-mentioned clauses are not satisfied,
a. For un-stiffened compression element, a reduction factor, $Q_s$, is introduced. Detailed values of $Q_s$ for different shapes are given in the clause XVII-2225.2 of NF-3000 1974.

b. For stiffened compression element, a reduced effective width, $b_e$, is introduced.

a. For the flanges of square and rectangular sections of uniform thickness:

\[ b_e = \frac{253t}{\sqrt{f}} \left[ 1 - \left( \frac{50.3}{b/t}\sqrt{f} \right) \right] \leq b \]

b. For other uniformly compressed elements:

\[ b_e = \frac{253t}{\sqrt{f}} \left[ 1 - \left( \frac{44.3}{b/t}\sqrt{f} \right) \right] \leq b \]

Consequently, a reduction factor, $Q_a$, equal to the effective area divided by the actual area is introduced. Combining both these factors, allowable stress for axially loaded compression members containing stiffened or un-stiffened elements shall not exceed

\[ F_a = F_y \left( \frac{Q_s Q_a \left[ 1 - \frac{(KL/r)^2}{2C'_c^2} \right]}{5 \left[ \frac{3(KL/r)}{8C'_c} \cdot \frac{(KL/r)^3}{8C'_c^3} \right]} \right) \]

Where:

\[ C'_c = \sqrt{\frac{2E}{Q_s Q_a F_y}} \]

D1L.1.1.4 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in XVII-2214 of NF-3000 1974 is:

a. Along Major Axis:

b. For Compact Sections, tension and compression on extreme fibers of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

\[ F_b = 0.66 \times F_y \]

If meeting the requirements of this member of:

a. Width-thickness ratio of un-stiffened projecting elements of the compression flange shall not exceed 52.2/$\sqrt{F_y}$.

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed 190/$\sqrt{F_y}$.

c. The depth-thickness ratio of the web shall not exceed

\[ \frac{d}{t} = \left( \frac{412}{\sqrt{F_y}} \right) \left[ 1 - 2.33\left( \frac{F_a}{F_y} \right) \right] \]

except that it need not be less than 257/$\sqrt{F_y}$.

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of $76b_F/\sqrt{F_y}$ nor $20000/(d/A_F)F_y$.

e. For noncompact and slender elements, clause XVII-2214.2 and XVII-2214.5 of NF-3000 1974 are followed respectively.

f. For box-type flexural members, maximum bending stress is:

\[ F_b = 0.60 \times F_y \]
Along Minor Axis:
For doubly symmetrical members (I shaped) meeting the requirements of XVII-2214.1(a) and (b) of NF-3000 1974, maximum tensile and compressive bending stress shall not exceed:

\[ F_b = 0.75 \times F_y \]

D1.L.1.5 Combined Interaction Check
Members subjected to both axial compression and bending stresses are proportioned to satisfy

\[ \frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{(1 - f_a/F_a')F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a/F_a')F_{by}} \leq 1.0 \]

and

\[ \frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0 \]

when \( f_a/F_a > 0.15 \), otherwise

\[ \frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0 \]

It should be noted that during code checking or member selection, if \( f_a/F_a \) exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient \( C_m \) is taken as 0.85 for side-sway and 0.6 - 0.4 \((M1/M2)\), but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy

\[ \frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0 \]

D1.L.1.6 Shear Stress
Allowable shear stress on the gross section [ref. XVII-2263.2 of NF-3000 1974] is calculated as

\[ F_v = (F_y/2.89)C_v \leq 0.4F_y \]

where

\[ C_v = \begin{cases} \frac{45,000k}{F_y(h/t)^2} & \text{when } h/b \leq 0.8 \\ \frac{190}{h/t} & \text{when } h/b > 0.8 \end{cases} \]

\[ k = \begin{cases} 4.00 + 5.34/(a/h)^2 & \text{when } a/h \leq 1.0 \\ 5.34 + 4.00/(a/h)^2 & \text{when } a/h > 1.0 \end{cases} \]

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.L.1.1 Member Property Specification
For specification of member properties, the specified steel section available in Steel Section Library of STAAD may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.
Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to [G.6 Member Properties](on page 2322).

### D1.L.1.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

**Note:** Unlike many other design codes available in STAAD.Pro (which use the BEAM parameter), design per ASME NF 3000 codes in STAAD.Pro is *always* performed based on forces calculated at 13 sections, including ends.

#### Table 116: ASME NF 3000 Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as CODE NF3000 1974 or CODE NF3000 1977. Design code to follow. See [TR.48.1 Parameter Specifications](on page 2851).</td>
</tr>
</tbody>
</table>
| CAN            | 0             | Used for Deflection Check only.  
0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2.  
1 = Deflection check based on the principle that maximum deflection is of the cantilever type |
| CB             | 1.0           | Bending coefficient dependent upon moment gradient.  
0.0 = CB is calculated itself  
Any other user-defined value is accepted. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CMY</td>
<td>0.85</td>
<td>Cm value in local y &amp; z axes</td>
</tr>
<tr>
<td>CMZ</td>
<td>for sidesway and calculated for no sidesway</td>
<td></td>
</tr>
<tr>
<td>CT</td>
<td>0.75</td>
<td>Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)]</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DMAX</td>
<td>45 inch</td>
<td>Maximum allowable depth. Used only with the MEMBER SELECTION command.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 inch</td>
<td>Minimum allowable depth. Used only with the MEMBER SELECTION command.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 KSI</td>
<td>Yield strength of steel at temperature in current units.</td>
</tr>
<tr>
<td>FU</td>
<td>60 KSI</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td>KBK</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit. Note: Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>KS</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses. Note: Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| KV             | 1.0           | Stress Limit Factor applicable to the Design Allowable Shear Stresses.  
**Note:** Ignored unless SRL is set to D. |
| KY             | 1.0           | K value in local y-axis. Usually, this is minor axis. |
| KZ             | 1.0           | K value in local z-axis. Usually, this is major axis. |
| LY             | Member Length | Length to calculate slenderness ratio for buckling about local Y axis. |
| LZ             | Member Length | Same as above except in z-axis (major). |
| MAIN           | 0.0           | 0.0 = check for slenderness  
1.0 = suppress slenderness check |
| NSF            | 1.0           | Net Section Factor for tension member. |
| PROFILE        | None          | Used in member selection. See TR, 48.1 Parameter Specifications (on page 2851) for details. |
| RATIO          | 1.0           | Permissible ratio of the actual to allowable stresses. |
| SRL            | A             | Service level, which defines the service level factors to use for modifying stress values for the service level conditions.  
A. Normal Conditions  
B. Upset  
C. Emergency  
D. Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined.  
Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1625) for additional information. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>STIFF</td>
<td>Member length or depth whichever is greater</td>
<td>Spacing of stiffeners for plate girder design</td>
</tr>
<tr>
<td>TMAIN</td>
<td>240 for main member 300 for “Truss” member</td>
<td>Slenderness limit under tension</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Controls the levels of detail to which results are reported.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Minimum detail</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Intermediate detail level</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Maximum detail</td>
</tr>
<tr>
<td>UNB</td>
<td>Member Length</td>
<td>Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.</td>
</tr>
<tr>
<td>UNT</td>
<td>Member Length</td>
<td>Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.</td>
</tr>
</tbody>
</table>

**Related Links**
- [D1.L.5. ASME NF 3000 Service Level Conditions](#) (on page 1625)

**D1.L.1.4 Code Checking and Member Selection**

Both code checking and member selection options are available with the ASME NF-3000 1974 and ASME NF-3000 1977 codes.

Refer to [D1.B.1.3 Code Checking](#) (on page 1418) for general information on Code Checking. Refer to [TR49 Code Checking Specification](#) (on page 2852) for details the specification of the Code Checking command.

Refer to [D1.B.1.4 Member Selection](#) (on page 1419) for general information on Member Selection. Refer to [TR.49.1 Member Selection Specification](#) (on page 2853) for details the specification of the Member Selection command.

**D1.L.1.5 Example**

A cantilever beam of length 30 inch is loaded at its free end with 5 kip compressive load and 5 kip lateral load. The beam is assigned with W24X104 steel member and is designed in accordance with ASME NF3000 1974.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-08
```
END JOB INFORMATION
UNIT INCHES KIP

JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;

DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03

END DEFINE MATERIAL

MEMBER PROPERTY AMERICAN
1 TABLE ST W24X104

CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1

JOINT LOAD
2 FX -5 FY -5

PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT JOINT DISPLACEMENTS
PRINT MEMBER FORCES

PARAMETER 1
CODE NF3000 1974
FYLD 36 ALL
FU 58 ALL
KY 0.9 ALL
KZ 0.9 ALL
NSF 0.85 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

The corresponding TRACK 2 output is as follows:

```
STAD.PRO CODE CHECKING - ( ASME NF3000-74) v2.0
******************************************************************************

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST W24X104 (AISC SECTIONS)
PASS NF-74-EQN-21 0.032 1
5.00 C 0.00 150.00 0.00
```

Design
D. Design Codes
D1.L.2. ASME NF 3000 - 1989 Code

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the American Society of Mechanical Engineers — Nuclear Facility (ASME NF) Code. The ASME NF-3000 1989 Code is used as the basis of this design.

A brief description of some of the major allowable stresses is described herein.
D1.L.2.1 Design Process

The design process follows the following design checks.

Each one of the checks are described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

D1.L.2.1.1 Slenderness

As per NF-3322.2(c), the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

D1.L.2.1.2 Tension

Allowable tensile stress on the Net section is calculated as \(0.60 \times F_y\), but not more than \(0.5 \times F_u\) on the Effective Net area.

The Net Area \(A_n\) shall be determined in accordance with NF-3322.8(c)(1) - (a), (b) and (c), and the NSF parameter can be utilized for that.

The Effective Net Area \(A_e\) of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. NF-3322.8(c)(1)(d)),

\[ A_e = C_t \times A_n \]

Unless otherwise specified, the default value of the CT parameter is set as 0.75.

The value of CT parameter for other conditions is described at section NF-3322.8(c)(1)(d)(1), (2) and (3).

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

D1.L.2.1.3 Compression

The allowable compressive stress for columns, except those fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(1). The allowable compressive stress for columns fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(2). The allowable compressive stress for member elements other than columns constructed by any material, including austenitic stainless steel, shall be as required by NF-3322.1(c) (3).

a. Gross Sections of Columns, except those fabricated of austenitic stainless steel:

   a. On gross section of axially loaded compression members, when \((KL/r) < C_c\),

   \[ F_a = F_y \left\{ \frac{1 - \frac{(KL/r)^2}{2C_c^2}}{3 + \frac{3(KL/r)}{8C_c} \cdot \frac{(KL/r)^3}{8C_c^3}} \right\} \]

   Where:

   \[ C_c = \sqrt{\frac{2a^2E}{F_y}} \]

   b. When \((KL/r) > C_c\),

   \[ F_a = \frac{12a^2E}{23(KL/r)^2} \]

   c. When \((KL/r) > 120\),
Design
D. Design Codes

\[ F_{as} = \frac{F_{y\text{ Eq.(a1) or Eq.(a2)}}}{1.6 \cdot \left(\frac{1}{200r}\right)} \]

b. Gross sections of columns fabricated from Austenitic Stainless steel:

a. When \((KL/r) \leq 120\),
\[ F_a = F_y \left(0.47 - \frac{KL}{444} \right) \]
b. When \((KL/r) > 120\),
\[ F_a = F_y \left(0.40 - \frac{KL}{600} \right) \]

c. Member elements other than columns:

a. For Plate Girder Stiffeners, \(F_a = 0.60 \cdot F_y\)
b. For webs of rolled shapes, \(F_a = 0.75 \cdot F_y\)

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the sub-sections of NF-3322.2(d).

If the above-mentioned clauses are not satisfied,

a. For un-stiffened compression element, a reduction factor, \(Q_a\), is introduced. Detailed values of \(Q_a\) for different shapes are given in NF-3322.2(e)(2)(a) to NF-3322.2(e)(2)(d).
b. For stiffened compression element, a reduced effective width, \(b_e\), is introduced.

a. For the flanges of square and rectangular sections of uniform thickness:
\[ b_e = \frac{253}{\sqrt{f}} \left[ 1 - \left( \frac{50.3}{b/t} \right) \right] \leq b \]
b. For other uniformly compressed elements:
\[ b_e = \frac{253}{\sqrt{f}} \left[ 1 - \left( \frac{44.3}{b/t} \right) \right] \leq b \]

Consequently, a reduction factor, \(Q_a\), equal to the effective area divided by the actual area is introduced.

Combining both these factors, allowable stress for axially loaded compression members containing stiffened or un-stiffened elements shall not exceed

\[ F_a = F_y \frac{Q_a Q_s}{5 \left[ 3 + \frac{3(KL/r)}{8C_c c} \cdot \left( \frac{KL/r}{c} \right)^3 \right]} \]

Where:
\[ C_c = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} \]

D1L.2.1.4 Bending Stress
Allowable bending stress for tension and compression for a structural member, as given in NF-3322.1(d) is:

A. Along Major Axis:
a. For Compact Sections, tension and compression on extreme fibres of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

\[ F_b = 0.66 \times F_y \]

If meeting the requirements of this member of:

a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed \( \frac{65}{\sqrt{F_y}} \).

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed \( \frac{190}{\sqrt{F_y}} \).

c. The depth-thickness ratio of the web shall not exceed

\[ \frac{d}{t} = \left( \frac{640}{\sqrt{F_y}} \right) \left[ 1 - 3.74 \left( \frac{f_a}{F_y} \right) \right] \text{ when } f_a/F_y \leq 0.16 \]

\[ \frac{d}{t} = \left( \frac{257}{\sqrt{F_y}} \right) \text{ when } f_a/F_y > 0.16 \]

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of \( \frac{76b_f}{\sqrt{F_y}} \) nor \( \frac{20000}{(d/A_f)F_y} \).

b. For noncompact and slender elements, NF-3322.1(d)(5) and NF-3322.1(d)(3) are followed respectively.

c. For box-type flexural members, maximum bending stress is:

\[ F_b = 0.60 \times F_y \]

**B. Along Minor Axis:**

a. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a) and (b), maximum tensile and compressive bending stress shall not exceed:

\[ F_b = 0.75 \times F_y \]

b. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a), except where \( b_f/2t_f \) exceeds \( \frac{65}{\sqrt{F_y}} \) but is less than \( \frac{95}{\sqrt{F_y}} \), maximum tensile and compressive bending stress shall not exceed:

\[ F_b = F_y \left[ 1.075 - 0.005 \left( \frac{b_f}{2t_f} \right) \sqrt{F_y} \right] \]

**D1.L.2.1.5 Combined Interaction Check**

Members subjected to both axial compression and bending stresses are proportioned to satisfy

\[ \frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{(1 - f_a/F_e)F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a/F_e)F_{by}} \leq 1.0 \]

and

\[ \frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0 \]

when \( f_a/F_a > 0.15 \), otherwise

\[ \frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0 \]

It should be noted that during code checking or member selection, if \( f_a/F_a \) exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient \( Cm \) is taken as 0.85 for side-sway and 0.6 - 0.4 \((M1/M2)\), but not less than 0.4 for no side-sway.
Members subjected to both axial tension and bending stress are proportioned to satisfy

\[
\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0
\]

D1.L.2.1.6 Shear Stress

Allowable shear stress on the gross section [ref. NF-3322.6(e)(2)] is calculated as

\[
F_v = \left(\frac{F_y}{2.89}\right) C_v \leq 0.4F_y
\]

where

\[
C_v = \begin{cases} 
\frac{45,000k}{F_y (h/t)^2} & \text{when } h/b \leq 0.8 \\
\frac{190}{h/t} \sqrt{\frac{k}{F_y}} & \text{when } h/b > 0.8 \\
4.00 + 5.34/(a/h)^2 & \text{when } a/h \leq 1.0 \\
5.34 + 4.00/(a/h)^2 & \text{when } a/h > 1.0 
\end{cases}
\]

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.L.2.2 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to G.6 Member Properties (on page 2322).

D1.L.2.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

Note: Unlike many other design codes available in STAAD.Pro (which use the BEAM parameter), design per ASME NF 3000 codes in STAAD.Pro is always performed based on forces calculated at 13 sections, including ends.
Table 117: ASME NF 3000 Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAN</td>
<td>0</td>
<td>Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Bending coefficient dependent upon moment gradient 0.0 = CB is calculated itself Any other user-defined value is accepted.</td>
</tr>
<tr>
<td>CT</td>
<td>0.75</td>
<td>Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)]</td>
</tr>
<tr>
<td>CMY CMZ</td>
<td>0.85 for sidesway and calculated for no sidesway</td>
<td>Cm value in local y &amp; z axes</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DMAX</td>
<td>4.5 inch</td>
<td>Maximum allowable depth, in current units. Used only with the MEMBER SELECTION command.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 inch</td>
<td>Minimum allowable depth, in current units. Used only with the MEMBER SELECTION command.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 KSI</td>
<td>Yield strength of steel at temperature in current units.</td>
</tr>
<tr>
<td>FU</td>
<td>60 KSI</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td>KBK</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit. <strong>Note:</strong> Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>KS</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses. <strong>Note:</strong> Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>KV</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Shear Stresses. <strong>Note:</strong> Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis. Usually, this is minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis. Usually, this is major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about local Y axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in z-axis (major).</td>
</tr>
</tbody>
</table>
| MAIN           | 0.0          | 0.0 = check for slenderness
1.0 = suppress slenderness check |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net Section Factor for tension member.</td>
</tr>
<tr>
<td>PROFILE</td>
<td>None</td>
<td>Used in member selection. See <a href="#">TR, 48.1 Parameter Specifications</a> (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
</tr>
<tr>
<td>SRL</td>
<td>A</td>
<td>Service level, which defines the service level factors to use for modifying stress values for the service level conditions.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A. Normal Conditions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>B. Upset</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C. Emergency</td>
</tr>
<tr>
<td></td>
<td></td>
<td>D. Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refer to <a href="#">D1.L.5. ASME NF 3000 Service Level Conditions</a> (on page 1625) for additional information.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Member length or depth whichever is greater</td>
<td>Spacing of stiffeners for plate girder design</td>
</tr>
<tr>
<td>STYPE</td>
<td>0.0</td>
<td>0.0 = Normal Steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = Austenitic Stainless Steel</td>
</tr>
<tr>
<td>TMAIN</td>
<td>240 for main member 300 for &quot;Truss&quot; member</td>
<td>Slenderness limit under tension</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Controls the levels of detail to which results are reported.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 = Minimum detail</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Intermediate detail level</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = Maximum detail</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>UNB</td>
<td>Member Length</td>
<td>Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.</td>
</tr>
<tr>
<td>UNT</td>
<td>Member Length</td>
<td>Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.</td>
</tr>
</tbody>
</table>

Related Links

- [D1.L.5. ASME NF 3000 Service Level Conditions](on page 1625)

**D1.L.2.4 Code Checking and Member Selection**

Both code checking and member selection options are available with the ASME NF-3000 1989 code.

Refer to [D1.B.1.3 Code Checking](on page 1418) for general information on Code Checking. Refer to [TR.49 Code Checking Specification](on page 2852) for details the specification of the Code Checking command.

Refer to [D1.B.1.4 Member Selection](on page 1419) for general information on Member Selection. Refer to [TR.49.1 Member Selection Specification](on page 2853) for details the specification of the Member Selection command.

**D1.L.2.5 Example**

A cantilever beam of length 100 inch is loaded at its free end with 5 kip compressive load and a uniformly distributed load of 1 kip/inch over the whole span. The beam is assigned with B571806 steel member and is designed in accordance with ASME NF3000 1989.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-08
END JOB INFORMATION
JOINT COORDINATES
1 0 0 0; 2 360 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST B571806
CONSTANTS
```
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FX -5
MEMBER LOAD
1 UNI GY -1.0 0 100
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PARAMETER 1
CODE NF3000 1989
STYLD 1 ALL
FYLD 36 ALL
KY 0.75 ALL
KZ 0.75 ALL
FU 58 ALL
NSF 0.9 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

The corresponding TRACK 2 output is as follows:

```
STAAD.PRO CODE CHECKING - ( ASME NF3000-89) v2.0
***********************************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST B571806 PASS SHEAR Y 0.770 1 5.00 C 0.00 5000.00 0.00
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SLENDERNESS CHECK: ACTUAL RATIO: 75.08 ALLOWABLE RATIO: 200.00</td>
</tr>
<tr>
<td>ALLOWABLE STRESSES: (UNIT - KIP INCH)</td>
</tr>
<tr>
<td>AXIAL: 1.13E+01 FCZ: 2.08E+01 FCY: 2.31E+01 FTZ: 2.16E+01 FTY: 2.31E+01</td>
</tr>
<tr>
<td>SHEAR: 5.18E+00</td>
</tr>
<tr>
<td>ACTUAL STRESSES: (UNIT - KIP INCH)</td>
</tr>
<tr>
<td>AXIAL: 1.06E-01 FBZ: 5.86E+00 FBY: 0.00E+00 SHEAR: 3.99E+00</td>
</tr>
<tr>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>SECTION PROPERTIES: (UNIT - INCH)</td>
</tr>
<tr>
<td>AXX: 47.00 AYY: 25.08 AZZ: 15.00 RZZ: 22.80 RYY: 3.60</td>
</tr>
<tr>
<td>SZZ: 853.77 SYY: 67.54</td>
</tr>
<tr>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>PARAMETER: (UNIT - KIP INCH)</td>
</tr>
<tr>
<td>KL/R-Z: 11.84 KL/R-Y: 75.08 UNL: 360.0 CMZ: 1.00 CMY: 1.00</td>
</tr>
<tr>
<td>CB: 1.75 FYLD: 36.00 FU: 58.00 NET SECTION FACTOR: 0.90</td>
</tr>
<tr>
<td>CT: 0.75 STEEL TYPE: 1.0 KS:1.000 KV:1.000 KBK:1.000</td>
</tr>
<tr>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -INCH)</td>
</tr>
</tbody>
</table>
```
### D1.L.3. ASME NF 3000 - 1998 Code

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the American Society of Mechanical Engineers — Nuclear Facility (ASME NF) Code. The ASME NF-3000 1998 Code is used as the basis of this design.

A brief description of some of the major allowable stresses is described herein.

#### D1.L.3.1 Design Process

The design process follows the following design checks.

Each one of the checks are described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

**D1.L.3.2.1 Slenderness**

As per NF-3322.2(c), the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

**D1.L.3.2.2 Tension**

Allowable tensile stress on the Net section is calculated as \((0.60\times F_y)\), but not more than \((0.5\times F_u)\) on the Effective Net area.

The Net Area \((A_n)\) shall be determined in accordance with NF-3322.8(c)(1)-(a), (b) and (c), and the NSF parameter can be utilized for that.

The Effective Net Area \((A_e)\) of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. NF-3322.8(c)(1)(d)),

\[
A_e = C_t \times A_n
\]

Unless otherwise specified, the default value of the CT parameter is set as 0.75.

The value of CT parameter for other conditions is described at section NF-3322.8(c)(1)(d)(1), (2) and (3).

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

**D1.L.3.2.3 Compression**

The allowable compressive stress for columns, except those fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(1). The allowable compressive stress for columns fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(2). The allowable compressive stress for member elements other than columns constructed by any material, including austenitic stainless steel, shall be as required by NF-3322.1(c)(3).

- **a. Gross Sections of Columns, except those fabricated of austenitic stainless steel:**
  - **a.** On gross section of axially loaded compression members, when \((KL/r) < C_c\),

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.009</td>
<td>1</td>
<td>5.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
<td>0.290</td>
<td>1</td>
<td>5.00E+00</td>
<td>-</td>
<td>5.00E+03</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
<td>0.000</td>
<td>1</td>
<td>5.00E+00</td>
<td>-</td>
<td>5.00E+03</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Y</td>
<td>0.770</td>
<td>1</td>
<td>-</td>
<td>1.00E+02</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
\[
F_a = F_y \left[ 1 - \left( \frac{KL}{r} \right)^2 \right]^{-\frac{1}{2} C_c^2}
\]

Where:
\[
C_c = \sqrt{\frac{2\pi^2 E}{F_y}}
\]

b. When \((KL/r) > C_c\),
\[
F_a = \frac{12\pi^2 E}{23(KL/r)^2}
\]
c. When \((KL/r) > 120\),
\[
F_{as} = \frac{F_a [Eq.\{a1\} or Eq.\{a2\}]}{1.6 \cdot \left( \frac{l}{200r} \right)}
\]

b. Gross sections of columns fabricated from Austenitic Stainless steel:

a. When \((KL/r) \leq 120\),
\[
F_a = F_y \left( 0.47 \cdot \frac{KL}{r} \right)
\]
b. When \((KL/r) > 120\),
\[
F_a = F_y \left( 0.40 \cdot \frac{KL}{r} \right)
\]
c. Member elements other than columns:

a. For Plate Girder Stiffeners, \(F_a = 0.60 \cdot F_y\)
b. For webs of rolled shapes, \(F_a = 0.75 \cdot F_y\)

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the sub-sections of NF-3322.2(d).

If the above-mentioned clauses are not satisfied,

a. For un-stiffened compression element, a reduction factor, \(Q_s\), is introduced. Detailed values of \(Q_s\) for different shapes are given in NF-3322.2(e)(2)(a) to NF-3322.2(e)(2)(d).

In the case for angles or plates projecting from compression members and for projecting elements of compression flanges of girder,

When \(95 / \sqrt{F_y / \frac{k_c}{F_y}} / k_c < b / t < 195 / \sqrt{F_y / \frac{k_c}{F_y}} / k_c\), \(Q_s = 1.293 \cdot 0.00309(b / t)\sqrt{F_y / k_c}\)

When \(b / t > 195 / \sqrt{F_y / k_c}\),
\[
Q_s = \frac{26, 200 k_c}{F_y (b / t)^2}
\]

Where:
\[
k_c = \frac{4.05}{(h / t)^{0.46}} \text{ when } h / t > 70, \text{ otherwise, } k_c = 1.0.
\]

b. For stiffened compression element, a reduced effective width, \(b_e\), is introduced.
a. For the flanges of square and rectangular sections of uniform thickness:

\[ b_e = \frac{253t}{\sqrt{f_y}} \left[ 1 - \left( \frac{50.3}{b/t} \right)^{1.5} \right] \leq b \]

b. For other uniformly compressed elements:

\[ b_e = \frac{253t}{\sqrt{f_y}} \left[ 1 - \left( \frac{44.3}{b/t} \right)^{1.5} \right] \leq b \]

Consequently, a reduction factor, \( Q_a \), equal to the effective area divided by the actual area is introduced. Combining both these factors, allowable stress for axially loaded compression members containing stiffened or un-stiffened elements shall not exceed

\[ F_a = F_y \frac{Q_s Q_a \left[ 1 - \frac{(KL/\rho)^2}{2C'c^2} \right]}{5 + 3\frac{(KL/\rho)}{8C'c} - \frac{(KL/\rho)^2}{8C'c^3}} \]

Where:

\[ C'c = \sqrt{\frac{2n^2E}{Q_s Q_a F_y}} \]

D1.L.3.2.4 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in NF-3322.1(d) is:

A. Along Major Axis:

1. For Compact Sections, tension and compression on extreme fibres of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

\[ F_b = 0.66 \times F_y \]

If meeting the requirements of this member of:

a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed \( 65/\sqrt{F_y} \).

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed \( 190/\sqrt{F_y} \).

c. The depth-thickness ratio of the web shall not exceed

\[ d/t = \frac{(640/\sqrt{F_y})[1 - 3.74(f_a/F_y)]}{f_a/F_y} \leq 0.16 \]

\[ d/t = \frac{257/\sqrt{F_y}}{2} \text{ when } f_a/F_y > 0.16 \]

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of \( 76b_f/\sqrt{F_y} \) nor \( 20,000/(d/A_f)F_y \).

2. For noncompact and slender elements, NF-3322.1(d)(5) and NF-3322.1(d)(3) are followed respectively.

3. For box-type flexural members, maximum bending stress is:

\[ F_b = 0.60 \times F_y \]

B. Along Minor Axis:

a. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a) and (b), maximum tensile and compressive bending stress shall not exceed:

\[ F_b = 0.75 \times F_y \]
b. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a), except where \( \frac{b_f}{2t_f} \) exceeds \( 65/\sqrt{F_y} \) but is less than \( 95/\sqrt{F_y} \), maximum tensile and compressive bending stress shall not exceed:

\[
F_b = F_y[1.075 - 0.005(b_f/2t_f)\sqrt{F_y}]
\]

D1.L.3.2.5 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy

\[
\frac{f_a}{F_a} + \frac{C_m f_{bz}}{(1 - f_a/F_a)F_{bx}} + \frac{C_m f_{by}}{(1 - f_a/F_a)F_{by}} \leq 1.0
\]

and

\[
\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0
\]

when \( f_a/F_a > 0.15 \), otherwise

\[
\frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0
\]

It should be noted that during code checking or member selection, if \( f_a/F_a \) exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient \( C_m \) is taken as 0.85 for side-sway and \( 0.6 - 0.4(M1/M2) \), but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy

\[
\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0
\]

D1.L.3.2.6 Shear Stress

Allowable shear stress on the gross section [ref. NF-3322.6(e)(2)] is calculated as

\[
F_v = (F_y/2.89)C_v \leq 0.4F_y
\]

where

\[
C_v = \begin{cases} 
\frac{45,000k}{F_y(h/t)^2} & \text{when } h/b \leq 0.8 \\
180h/t & \text{when } h/b > 0.8 \\
4.00 + 5.34/(a/h)^2 & \text{when } a/h \leq 1.0 \\
5.34 + 4.00/(a/h)^2 & \text{when } a/h > 1.0
\end{cases}
\]

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.L.3.3 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD.Pro may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.
For more information on these facilities, refer to G.6 Member Properties (on page 2322).

**D1.L.3.4 Design Parameters**

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

**Note:** Unlike many other design codes available in STAAD.Pro (which use the BEAM parameter), design per ASME NF 3000 codes in STAAD.Pro is always performed based on forces calculated at 13 sections, including ends.

Table 118: ASME NF 3000 1998 Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAN</td>
<td>0</td>
<td>Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Bending coefficient dependent upon moment gradient 0.0 = CB is calculated itself Any other user-defined value is accepted.</td>
</tr>
<tr>
<td>CMY CMZ</td>
<td>0.85 for sidesway and calculated for no sidesway</td>
<td>Cm value in local y &amp; z axes</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CT</td>
<td>0.75</td>
<td>Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)]</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot;</td>
</tr>
<tr>
<td>DMAX</td>
<td>45 inch</td>
<td>Maximum allowable depth, in current units. Used only with the MEMBER SELECTION command.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 inch</td>
<td>Minimum allowable depth, in current units. Used only with the MEMBER SELECTION command.</td>
</tr>
<tr>
<td>FYLD</td>
<td>36 KSI</td>
<td>Yield strength of steel at temperature in current units.</td>
</tr>
<tr>
<td>FU</td>
<td>60 KSI</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td>KBK</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending Stresses to determine the Buckling Limit.</td>
</tr>
<tr>
<td>KS</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses.</td>
</tr>
<tr>
<td>KV</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Shear Stresses.</td>
</tr>
</tbody>
</table>

**Note:** Ignored unless SRL is set to D.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis. Usually, this is minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis. Usually, this is major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about local Y axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in z-axis (major).</td>
</tr>
<tr>
<td>MAIN</td>
<td>0.0</td>
<td>0.0 = check for slenderness check 1.0 = suppress slenderness check</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net Section Factor for tension member.</td>
</tr>
<tr>
<td>PROFILE</td>
<td>None</td>
<td>Used in member selection. See TR. 48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
</tr>
<tr>
<td>SRL</td>
<td>A</td>
<td>Service level, which defines the service level factors to use for modifying stress values for the service level conditions.</td>
</tr>
</tbody>
</table>
|                |               | A. Normal Conditions  
|                |               | B. Upset  
|                |               | C. Emergency  
|                |               | D. Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined.  
|                |               | Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1625) for additional information. |
| STIFF          | Member length or depth whichever is greater | Spacing of stiffeners for plate girder design |
| STYPE          | 0.0           | 0.0 = Normal Steel  
|                |               | 1.0 = Austenitic Stainless Steel |
### Design

#### D. Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| TMAIN          | 240 for main member  
               | 300 for "Truss" member | Slenderness limit under tension |
| TRACK          | 0.0           | Controls the levels of detail to which results are reported.  
                | 0 = Minimum detail  
                | 1 = Intermediate detail level  
                | 2 = Maximum detail |
| UNB            | Member Length | Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange. |
| UNT            | Member Length | Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange. |

**Notes**

1. All values are entered in the current units.
2. The parameters DMAX and DMIN are only used with the MEMBER SELECTION command.

**Related Links**

- [D1.L.5. ASME NF 3000 Service Level Conditions](#) (on page 1625)

**D1.L.3.5 Code Checking and Member Selection**

Both code checking and member selection options are available with the ASME NF-3000 1998 code.

Refer to [D1.B.1.3 Code Checking](#) (on page 1418) for general information on Code Checking. Refer to [TR.49 Code Checking Specification](#) (on page 2852) for details the specification of the Code Checking command.

Refer to [D1.B.1.4 Member Selection](#) (on page 1419) for general information on Member Selection. Refer to [TR.49.1 Member Selection Specification](#) (on page 2853) for details the specification of the Member Selection command.

**D1.L.3.6 Example**

A cantilever beam of length 100 inch is loaded at its free end with 5 kip compressive load and a uniformly distributed load of 1 kip/inch over the whole span. The beam is assigned with B571806 steel member and is designed in accordance with ASME NF3000 1998.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
```
The corresponding TRACK 2 output is as follows:

```
STAAD.PRO CODE CHECKING - ( ASME NF3000-98) v2.0
********************************************************************************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER    TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
FX            MY             MZ       LOCATION
=======================================================================
1  ST   B571806                  (AISC SECTIONS)  PASS      SHEAR Y           0.635         1
5.00 C          0.00        5000.00        0.00
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SLENDERNESS CHECK:    ACTUAL RATIO:  20.85 ALLOWABLE RATIO: 200.00</td>
</tr>
<tr>
<td>ALLOWABLE STRESSES:   (UNIT - KIP  INCH)</td>
</tr>
</tbody>
</table>
********************************************************************************
```
D1.L.4. ASME NF 3000 - 2001 & 2004 Codes

STAAD.Pro is capable of performing steel design based on the American Society of Mechanical Engineers Nuclear Facility Code, ASME NF 3000 - 2004.

**Note:** Use of 2004 edition requires STAAD.Pro V8i (SELECTseries 2) NRC (build 20.07.07.30) or higher. Use of 2001 edition requires STAAD.Pro V8i (SELECTseries 3) NRC (build 20.07.08.22) or higher.

Design of members per ASME NF 3000 - 2001 & 2004 requires the **STAAD Nuclear Design Codes** SELECT Code Pack.

**D1.L.4.1 Design Process**

The design process follows the following design checks.

Each one of the checks is described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

**D1.L.4.1.1 Slenderness**

As per NF-3322.2(c), the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

**D1.L.4.1.2 Tension**

Allowable tensile stress on the Net section is calculated as \(0.60 \times F_y\), but not more than \(0.5 \times F_u\) on the Effective Net area.

The Net Area \(A_n\) shall be determined in accordance with NF-3322.8(c)(1) - (a), (b) and (c), and the NSF parameter can be utilized for that.

The Effective Net Area \(A_o\) of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. NF-3322.8(c)(1)(d)).

\[
A_o = C_t \times A_n
\]

Unless otherwise specified, the default value of the CT parameter is set as 0.75.
The value of CT parameter for other conditions is described at section NF-3322.8(c)(1)(d)(1), (2) and (3).

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

D1.4.1.3 Compression

The allowable compressive stress for columns, except those fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(1). The allowable compressive stress for columns fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(2). The allowable compressive stress for member elements other than columns constructed by any material, including austenitic stainless steel, shall be as required by NF-3322.1(c)(3).

A. Gross Sections of Columns, except those fabricated of austenitic stainless steel:

a. On gross section of axially loaded compression members, when \( (KL/r) < C_c \),

\[
F_a = F_y \left[ 1 + \frac{1}{2} \left( \frac{KL}{r} \right)^2 \frac{2C_c^2}{2C_c^2} \right]^{-3} + \frac{3}{8C_c} \left( \frac{KL}{r} \right)^3
\]

Where:

\[
C_c = \sqrt{\frac{2\pi^2 E}{F_y}}
\]

b. When \( (KL/r) > C_c \),

\[
F_a = \frac{12\pi^2 E}{23(KL/r)^2}
\]

c. When \( (KL/r) > 120 \),

\[
F_{as} = \frac{F_y [Eq.(a1) or Eq.(a2)]}{1.6 \cdot \left( \frac{1}{200r} \right)}
\]

B. Gross sections of columns fabricated from Austenitic Stainless steel:

a. When \( (KL/r) \leq 120 \),

\[
F_a = F_y [0.47 \cdot (KL/r)/444]
\]

b. When \( (KL/r) > 120 \),

\[
F_a = F_y [0.40 \cdot (KL/r)/600]
\]

C. Member elements other than columns:

a. For Plate Girder Stiffeners,

\[
F_a = 0.60 \cdot F_y
\]

b. For webs of rolled shapes,

\[
F_a = 0.75 \cdot F_y
\]

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the subsections of NF-3322.2(d).

If the above-mentioned clauses are not satisfied,

a. For un-stiffened compression element,
A reduction factor $Q_s$ is introduced. Detailed values of $Q_s$ for different shapes are given in NF-3322.2(e)(2)(a) to NF-3322.2(e)(2)(d).

In the case for angles or plates projecting from compression members and for projecting elements of compression flanges of girder,

When $95/(F_y/kc)^{1/2} < b/t < 195/(F_y/kc)^{1/2}$, $Q_s = 1.293 \cdot 0.00309 \cdot (b/t) \cdot (F_y/kc)^{1/2}$

When $b/t > 195/(F_y/kc)^{1/2}$, $Q_s = 26,200 \cdot kc/[F_y(b/t)^2]$

Where:

$k_c = 4.05/[(h/t)^{0.46}]$ if $h/t > 70$, otherwise $k_c = 1.0$.

b. For stiffened compression element,

A reduced effective width $b_e$ is introduced.

a. For the flanges of square and rectangular sections of uniform thickness:

$$b_e = \frac{253}{\sqrt{f}} \left[ 1 - \left( \frac{50.3}{(b/t)\sqrt{f}} \right) \right] \leq b$$

b. For other uniformly compressed elements:

$$b_e = \frac{253}{\sqrt{f}} \left[ 1 - \left( \frac{44.3}{(b/t)\sqrt{f}} \right) \right] \leq b$$

Consequently, a reduction factor $Q_a$ is introduced and is equal to the effective area divided by the actual area. Combining both these factors, allowable stress for axially loaded compression members containing stiffened or unstiffened elements shall not exceed

$$F_a = F_y \frac{Q_s Q_a \left[ 1 \cdot \frac{(KL/r)^2}{2c^2} \right]}{5 + 3 \frac{(KL/r)}{8c^2} \cdot \frac{(KL/r)^3}{2c^2}}$$

Where:

$$C' = \sqrt{\frac{2n^2 E}{Q_s Q_a F_y}}$$

D1.L.4.1.4 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in NF-3322.1(d) is:

A. Along Major Axis:

a. For Compact Sections, tension and compression on extreme fibres of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

$$F_b = 0.66 \times F_y$$

If meeting the requirements of this member of:

a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed $65/\sqrt{F_y}$.

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed $190/\sqrt{F_y}$.

c. The depth-thickness ratio of the web shall not exceed

$$d/t = (640/\sqrt{F_y})(1 \cdot 3.74(f_a/F_y))$$ when $f_a/F_y \leq 0.16$
d/t = 257/√Fy when fa/Fy > 0.16

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of 76bf/√Fy nor 20,000/(d/Af)Fy.

b. For noncompact and slender elements, NF-3322.1(d)(5) and NF-3322.1(d)(3) are followed respectively.

c. For box-type flexural members, maximum bending stress is:

\[ F_b = 0.60 \times F_y \]

B. Along Minor Axis:

a. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a) and (b), maximum tensile and compressive bending stress shall not exceed:

\[ F_b = 0.75 \times F_y \]

b. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a), except where bf/2tf exceeds 65/√Fy but is less than 95/√Fy, maximum tensile and compressive bending stress shall not exceed:

\[ F_b = F_y[1.075 \times 0.005(bf/2tf)^{\frac{1}{2}}] \]

D1.L.4.1.5 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy

\[
\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{(1 - f_a/F'_ex) F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a/F'_ey) F_{by}} \leq 1.0
\]

and

\[
\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0
\]

when \( f_a/F_a > 0.15 \), otherwise

\[
\frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0
\]

It should be noted that during code checking or member selection, if \( f_a/F_a \) exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient \( C_m \) is taken as 0.85 for side-sway and \( 0.6 \times 0.4 \times (M1/M2) \), but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy

\[
\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0
\]

D1.L.4.1.6 Shear Stress

Allowable shear stress on the gross section [ref. NF-3322.6(e)(2)] is calculated as

\[ F_v = (F_y/2.89) C_v \leq 0.4F_y \]

where

\[
C_v = \begin{cases} 
45,000k 
& \text{when } h/b \leq 0.8 \\
F_y(h/t)^2 
& \text{when } h/b > 0.8 \\
\end{cases}
\]

\[
k = \begin{cases} 
4.00 + 5.34/(a/h)^2 
& \text{when } a/h \leq 1.0 \\
& \text{when } a/h > 1.0 \\
\end{cases}
\]
\[ 5.34 \times \frac{a}{h} + 4.00 \left( \frac{a}{h} \right)^2 \text{ when } a/h > 1.0 \]

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

**D1.L.4.2 Member Property Specification**

For specification of member properties, the specified steel section available in Steel Section Library of STAAD.Pro may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to [G.6 Member Properties](#) (on page 2322).

**D1.L.4.3 Design Parameters**

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

**Note:** Unlike many other design codes available in STAAD.Pro (which use the BEAM parameter), design per ASME NF 3000 codes in STAAD.Pro is always performed based on forces calculated at 13 sections, including ends.

**Table 119: ASME NF 3000 2001 & 2004 Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as NF3000 2001 or NF3000 2004. Specified design code is followed for code checking purpose. Design code to follow. See <a href="#">TR48.1 Parameter Specifications</a> (on page 2851).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| CAN            | 0             | Used for Deflection Check only.  
  0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2.  
  1 = Deflection check based on the principle that maximum deflection is of the cantilever type |
| CB             | 1.0           | Bending coefficient dependent upon moment gradient  
  0.0 = CB is calculated itself  
  Any other user-defined value is accepted. |
<p>| CMY CMZ        | 0.85          | Cm value in local y &amp; z axes |
|                |               | 0.85 for sidesway and calculated for no sidesway |
| CT             | 0.75          | Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)] |
| DFF            | None (Mandatory for deflection check) | &quot;Deflection Length&quot; / Maximum allowable local deflection |
| DJ1            | Start Joint of the member | Joint No. denoting starting point for calculation of &quot;Deflection Length&quot; |
| DJ2            | End Joint of the member | Joint No. denoting end point for calculation of &quot;Deflection Length&quot; |
| DMAX           | 45 inch       | Maximum allowable depth |
|DMIN           | 0.0 inch      | Minimum allowable depth |
| FYLD           | 36 KSI        | Yield strength of steel at temperature in current units. |
| FU             | 60 KSI        | Ultimate tensile strength of steel in current units. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>KBK</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit. &lt;br&gt;&lt;br&gt;Note: Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>KS</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses. &lt;br&gt;&lt;br&gt;Note: Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>KV</td>
<td>1.0</td>
<td>Stress Limit Factor applicable to the Design Allowable Shear Stresses. &lt;br&gt;&lt;br&gt;Note: Ignored unless SRL is set to D.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis. Usually, this is minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis. Usually, this is major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for buckling about local Y axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in z-axis (major).</td>
</tr>
<tr>
<td>MAIN</td>
<td>0.0</td>
<td>0.0 = check for slenderness &lt;br&gt;1.0 = suppress slenderness check</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net Section Factor for tension member.</td>
</tr>
<tr>
<td>PROFILE</td>
<td>None</td>
<td>Used in member selection. See TR, 48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SRL</td>
<td>A</td>
<td>Service level, which defines the service level factors to use for modifying stress values for the service level conditions. A. Normal Conditions B. Upset C. Emergency D. Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined. Refer to D1.1.5, ASME NF 3000 Service Level Conditions (on page 1625) for additional information.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Member length or depth whichever is greater</td>
<td>Spacing of stiffeners for plate girder design</td>
</tr>
<tr>
<td>STYPE</td>
<td>0.0</td>
<td>0.0 = Normal Steel 1.0 = Austenitic Stainless Steel</td>
</tr>
<tr>
<td>TMAIN</td>
<td>240 for main member 300 for &quot;Truss&quot; member</td>
<td>Slenderness limit under tension</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Controls the levels of detail to which results are reported. • Minimum detail • Intermediate detail level • Maximum detail</td>
</tr>
<tr>
<td>UNB</td>
<td>Member Length</td>
<td>Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.</td>
</tr>
<tr>
<td>UNT</td>
<td>Member Length</td>
<td>Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.</td>
</tr>
</tbody>
</table>

Notes

1. All values are entered in the current units.
2. The parameters DMAX and DMIN are only used with the MEMBER SELECTION command.

Related Links
- **D1.L.5. ASME NF 3000 Service Level Conditions** (on page 1625)

**D1.L.4.4 Code Checking and Member Selection**

Both code checking and member selection options are available with the ASME NF-3000 2004 code.

Refer to **D1.B.1.3 Code Checking** (on page 1418) for general information on Code Checking. Refer to **TR49 Code Checking Specification** (on page 2852) for details the specification of the Code Checking command.

Refer to **D1.B.1.4 Member Selection** (on page 1419) for general information on Member Selection. Refer to **TR.49.1 Member Selection Specification** (on page 2853) for details the specification of the Member Selection command.

**D1.L.4.5 Example of 2004 Code**

A cantilever beam of length 100 inch is loaded at its free end with 5 kip compressive load and a uniformly distributed load of 1 kip/inch over the whole span. The beam is assigned with B571806 steel member and is designed in accordance with ASME NF3000 2004.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-08
END JOB INFORMATION
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 100 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST B571806
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FX -5
MEMBER LOAD
1 UNI GY -1.0 0 100
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
STYPE 1 ALL
FYLD 36 ALL
KY 0.75 ALL
```
KZ 0.75 ALL
FU 58 ALL
NSF 0.9 ALL
CT 0.85 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

The corresponding TRACK 2 output is as follows:

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/CRITICAL COND/RATIO/LOADING/LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX  MY  MZ  LOAD</td>
</tr>
<tr>
<td>1</td>
<td>ST</td>
<td>PASS  NF-3322.1(b)  0.635  1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5.00 C  0.00  5000.00  0.00</td>
</tr>
</tbody>
</table>

SLENDERNESS CHECK: ACTUAL RATIO: 20.85 ALLOWABLE RATIO: 200.00
ALLOWABLE STRESSES: (UNIT - KIP INCH)
AXIAL: 1.20E+01  FCZ: 2.22E+01  FCY: 2.31E+01  FTZ: 2.22E+01  FTY: 2.31E+01
SHEAR: 6.28E+00
ACTUAL STRESSES: (UNIT - KIP INCH)
AXIAL: 1.06E-01  FBZ: 5.86E+00  FBY: 0.00E+00  SHEAR: 3.99E+00

SECTION PROPERTIES: (UNIT - INCH)
Axx: 47.00  Ayy: 25.08  Azz: 15.00  Rzz: 22.80  Ryy: 3.60
Szz: 853.77  Syy: 67.54

PARAMETER: (UNIT - KIP INCH)
KL/R-Z: 3.29  KL/R-Y: 20.85  UNL: 100.0  CMZ: 1.00  CMY: 1.00
CB: 1.75  FYLD: 36.00  FU: 58.00  NET SECTION FACTOR: 0.90
CT: 0.85  STEEL TYPE: 1.0  KS:1.000  KV:1.000  KBK:1.000

CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -INCH)
<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.005</td>
<td>1</td>
<td>5.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.000</td>
<td>1</td>
<td>5.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMB&amp;END</td>
<td>0.272</td>
<td>1</td>
<td>5.00E+00</td>
<td>-</td>
<td>-</td>
<td>5.00E+03</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
<td>0.000</td>
<td>1</td>
<td>5.00E+00</td>
<td>-</td>
<td>-</td>
<td>5.00E+03</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>SHEAR-Y</td>
<td>0.635</td>
<td>1</td>
<td>-</td>
<td>1.00E+02</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Z</td>
<td>0.000</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

**Note:** An asterisk following a critical load case number indicates that this load case is a generated load combination. See TR.35 Load Combination Specification (on page 2791) for additional information.

**D1.1.5. ASME NF 3000 Service Level Conditions**

Service Level Conditions are basically the loading conditions for which the plant structure and its components are to be designed. The same primary load can be multiplied by different factors to signify the different service levels. Also the load combinations for various service levels are different and pre-defined by the code.
**D1.L.5.1 Service Levels**

The following is a short overview of each of the service levels specified by the code:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A. Normal Working</td>
<td>This situation can be termed as a short term failure or a local failure, and the repairing or modification of the structure can be done without shutting the entire plant.</td>
</tr>
<tr>
<td>B. Upset</td>
<td>This situation can be termed as a major failure, and the repairing of the structure can be done only after shutting down the entire plant.</td>
</tr>
<tr>
<td>C. Emergency</td>
<td>This situation can be termed as devastation, and the main objective of this level is to have sufficient time for safe relocation of human life and valuable properties, and to initiate the controlled failure of the plant structure. The plant is already at an unusable state and a rare chance to repair it back into the operation.</td>
</tr>
<tr>
<td>D. Faulted</td>
<td></td>
</tr>
</tbody>
</table>

These Service Levels are the attribute of the whole structure or the structural system. So, the existence of different Service Levels to the different parts of the structure at the same point of time is totally ruled out.

The Service Level Factors are basically few multiplying factor by which the Allowable Stress values are to be multiplied based on the Service Level. The different actions (e.g., Tension, Compression, Bending, Shear etc.) have different Service Level Factors.

However, this is to be noted, the stipulated multiplying factors for creating load combinations for Service Level B, C, and D are to be user defined in this case. The facility of creating Auto Load Combination for different Service Levels is out of the scope of this implementation. The user has to take care of this.

**D1.L.5.2 Stress Level Factors**

For the Member Design, as per Clause NF-3321.1, the Allowable Stresses may be increased by the Factors as per Table NF-3523(b)-1 and NF-3623(b)-1. Table NF-3523(b)-1 is applicable to Component Support Structures and Table NF-3623(b)-1 is applicable to Piping Support Structures. However, as the values are the same for the service level factors in each table, STAAD.Pro does not make any differentiation between component and piping supports.

**Note:** Clause NF-3321.1 also indicates that the allowable stress shall be limited to two-thirds (2/3x) the critical buckling stress. However, the critical buckling stress is not clearly defined so it is left to the user to ensure that this code requirement is met.

The values used for the stress level factors in STAAD.Pro are as follows:

<table>
<thead>
<tr>
<th>Service Level</th>
<th>Ks</th>
<th>Kv</th>
<th>Kbk</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>Service Level</td>
<td>Ks</td>
<td>Kv</td>
<td>Kbk</td>
</tr>
<tr>
<td>---------------</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
</tr>
<tr>
<td>B</td>
<td>1.33</td>
<td>1.33</td>
<td>1.33</td>
</tr>
<tr>
<td>C</td>
<td>1.50</td>
<td>1.50</td>
<td>1.50</td>
</tr>
<tr>
<td>D*</td>
<td>KS</td>
<td>KV</td>
<td>KBK</td>
</tr>
</tbody>
</table>

* It is evident from the Table NF-3523(b)-1, that there are no predefined Stress Limit Factors for Service Level D. So, for Service Level D, the Factors Ks, Kv and Kbk are to be user defined. Refer to Appendix F in the code for guidance on values to specify in the design parameters.

where

\( K_s \)  
Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses.

\( K_v \)  
Stress Limit Factor applicable to the Design Allowable Shear Stresses

\( K_{bk} \)  
Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit.

The program uses the service level factors —either those specified for levels A through C or the user defined values in level D— as follows:

- The Allowable Axial Tensile Stress is to be multiplied by \( K_s \)
- The Allowable Axial Compressive Stress is to be multiplied by \( K_{bk} \)
- The Allowable Bending Stress is to be multiplied by \( K_s \)
- The Allowable Shear Stress is to be multiplied by \( K_v \)
- As per NF-3322.1(e), for checking Combined Stresses as per equation 20, the value of \( F'_e \) and \( F'_{ez} \) — the Euler Stress divided by the factor of safety, may also be multiplied by the appropriate Stress Limit Factor. This is also implemented. \( F'e \) is to be multiplied by \( K_{bk} \).

Related Links

- D1.L.1.3 Design Parameters (on page 1592)
- D1.L.2.3 Design Parameters (on page 1601)
- D1.L.3.4 Design Parameters (on page 1611)
- D1.L.4.3 Design Parameters (on page 1620)

D2. Australian Codes

D2.A. Australian Codes - Concrete Design per AS 3600 - 2001

STAAD.Pro is capable of performing concrete design based on the Australian code AS 3600-2001 Australian Standard-Concrete Structures.

D2.A.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams: Prismatic (Rectangular & Square)
- For Columns: Prismatic (Rectangular, Square, and Circular)
D2.A.2 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the MEMBER PROPERTY command. The following example shows the required input:

UNIT MM
MEMBER PROPERTY
1 3 TO 7 9 PRISM YD 450. ZD 250.
11 13 PR YD 350.

In the above input, the first set of members are rectangular (450 mm depth and 250mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 350 mm diameter. It is absolutely imperative that the user not provide the cross section area (AX) as an input.

D2.A.3 Design Parameters

The program contains a number of parameters which are needed to perform the design. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 1A.1 of this manual contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 120: Australian Concrete Design per AS 3600 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td></td>
<td>Must be specified as AUSTRALIAN to invokes design per AS 3600 - 2001. Design code to follow. See [TR.53.2 Concrete Design-Parameter Specification](on page 2859).</td>
</tr>
<tr>
<td>CLB</td>
<td>40 mm</td>
<td>Clear cover for outermost bottom reinforcement.</td>
</tr>
<tr>
<td>CLS</td>
<td>40 mm</td>
<td>Clear cover for outermost side reinforcement.</td>
</tr>
<tr>
<td>CLT</td>
<td>40 mm</td>
<td>Clear cover for outermost top reinforcement.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>Total depth to be used for design. This value defaults to YD as provided under MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>EFACE</td>
<td>0</td>
<td>Distance from end node of beam to face of support used for shear design; used for shear and torsion calculations.</td>
</tr>
<tr>
<td>FC</td>
<td>30 N/mm²</td>
<td>Compressive strength of concrete.</td>
</tr>
<tr>
<td>FYMAIN</td>
<td>400 N/mm²</td>
<td>Yield Stress for main reinforcing steel. Applicable values per Table 6.2.1 of AS 3600-2001: 250, 400, 450, 500</td>
</tr>
<tr>
<td>FYSEC</td>
<td>400 N/mm²</td>
<td>Yield Stress for secondary reinforcing steel. Applicable values per Table 6.2.1 of AS 3600-2001: 250, 400, 450, 500</td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>12 mm</td>
<td>Maximum main reinforcement bar size.</td>
</tr>
<tr>
<td>MINMAIN</td>
<td>12 mm</td>
<td>Minimum main reinforcement bar size.</td>
</tr>
<tr>
<td>MINSEC</td>
<td>12 mm</td>
<td>Minimum secondary reinforcement bar size.</td>
</tr>
<tr>
<td>MMAG</td>
<td>1</td>
<td>Factor by which column design moments are magnified.</td>
</tr>
<tr>
<td>RATIO</td>
<td>4.0</td>
<td>Maximum percentage of longitudinal reinforcement in columns.</td>
</tr>
<tr>
<td>NSECTION</td>
<td>12</td>
<td>Number of equally spaced sections for design.</td>
</tr>
<tr>
<td>SFACE</td>
<td>0</td>
<td>Distance from start node of beam to face of support used for shear design; used for shear and torsion calculations.</td>
</tr>
</tbody>
</table>
### Parameter Name | Default Value | Description
--- | --- | ---
| TRACK | 0.0 | For beam design:
- 0.0 = output consists of reinforcement details at the member start, middle, and end
- 1.0 = critical moments are printed in addition to TRACK 0.0 output
- 2.0 = required steel for intermediate sections defined by NSECTION are printed in addition to TRACK 0.0 output

For column design:
- 0.0 = reinforcement details are printed

| WIDTH | ZD | Width to be used for design. This value defaults to ZD as provided under MEMBER PROPERTIES.

---

**D2.A.4 Slenderness Effects and Analysis Consideration**

Slenderness effects are extremely important in designing compression members. There are two options by which the slenderness effect can be accommodated. One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed end moments, the effect of deflections on moment and forces and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD.Pro has been written to allow the use of the first option. To perform this type of analysis, use the command **PDELTA ANALYSIS** instead of **PERFORM ANALYSIS**. The **PDELTA ANALYSIS** will accommodate the requirements of the second-order analysis described by AS 3600, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the evaluation of slenderness effects is also by an approximate method. In this method, additional moments are calculated based on empirical formula and assumptions on sidesway.

Considering all of the above information, a P-Delta analysis —as performed by STAAD— may be used for the design of concrete members. However the user must note that to take advantage of this analysis, all the combinations of loading must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a primary load case is revised during the P-delta analysis based on the deflections. Also, note that the proper factored loads (like 1.5 for dead load etc.) should be provided by the user. STAAD.Pro does not factor the loads automatically.
D2.A.5 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The total number of sections considered is 13 (e.g., 0., .1, .2, .25, .3, .4, .5, .6, .7, .75, .8, .9, and 1). All of these sections are scanned to determine the design force envelopes.

D2.A.5.1 Design for Flexure

Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the above mentioned sections. Each of these sections is designed to resist both of these critical sagging and hogging moments. Currently, design of singly reinforced sections only is permitted. If the section dimensions are inadequate as a singly reinforced section, such a message will be permitted in the output. Flexural design of beams is performed in two passes. In the first pass, effective depths of the sections are determined with the assumption of single layer of assumed reinforcement and reinforcement requirements are calculated. After the preliminary design, reinforcing bars are chosen from the internal database in single or multiple layers. The entire flexure design is performed again in a second pass taking into account the changed effective depths of sections calculated on the basis of reinforcement provided after the preliminary design. Final provisions of flexural reinforcements are made then. Efforts have been made to meet the guideline for the curtailment of reinforcements as per AS 3600. Although exact curtailment lengths are not mentioned explicitly in the design output (finally which will be more or less guided by the detailer taking into account of other practical consideration), user has the choice of printing reinforcements provided by STAAD at 13 equally spaced sections from which the final detailed drawing can be prepared.

D2.A.5.2 Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear design is performed at 13 equally spaced sections (0. to 1.) for the maximum shear forces amongst the active load cases and the associated torsional moments. Shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by STAAD. Two-legged stirrups are provided to take care of the balance shear forces acting on these sections.

Example of Input Data for Beam Design:

```
UNIT NEWTON MMS
START CONCRETE DESIGN
CODE AUSTRALIAN
FYMAIN 415 ALL
FYSEC 415 ALL
FC 35 ALL
CLEAR 25 MEM 2 TO 6
MAXMAIN 40 MEMB 2 TO 6
TRACK 1.0 MEMB 2 TO 9
DESIGN BEAM 2 TO 9
END CONCRETE DESIGN
```

D2.A.6 Column Design

Columns are designed for axial forces and biaxial moments at the ends. All active load cases are tested to calculate reinforcement. The loading which yields maximum reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. By default, square and rectangular columns are
designed with reinforcement distributed on each side equally. That means the total number of bars will always be a multiple of four (4). This may cause slightly conservative results in some cases. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by AS 3600 have been taken care of in the column design of STAAD.

```
Example of Input Data for Column Design:
UNIT NEWTON MMS
START CONCRETE DESIGN
CODE AUSTRALIAN
FYMAIN 415 ALL
FC 35 ALL
CLEAR 25 MEMB 2 TO 6
MAXMAIN 40 MEMB 2 TO 6
DESIGN COLUMN 2 TO 6
END CONCRETE DESIGN
```

D2.A.7 Slab or Wall Design

To design a slab or wall, it must be modeled using finite elements. The command specifications are in accordance with Chapter 2 and Chapter 6 of the specification.

Elements are designed for the moments $M_x$ and $M_y$. These moments are obtained from the element force output. The reinforcement required to resist $M_x$ moment is denoted as longitudinal reinforcement and the reinforcement required to resist $M_y$ moment is denoted as transverse reinforcement. The parameters $FYMAIN$, $FC$, $MAXMAIN$, $MINMAIN$, and $CLEAR$ listed in D2.A.3 Design Parameters (on page 1628) are relevant to slab design. Other parameters mentioned in Table 1A.1 are not applicable to slab design.

```
Example of Input Data for Slab/Wall Design
UNIT NEWTON MMS
START CONCRETE DESIGN
CODE AUSTRALIAN
FYMAIN 415 ALL
FC 25 ALL
```

*Figure 173: Element moments: Longitudinal (L) and Transverse (T)*
D2.B. Australian Codes - Steel Design per AS 4100 - 1998

STAAD.Pro is capable of performing steel design based on the Australian code AS 4100-1998 Standards Australia - Steel Structural Design, including Amendment 1 (2012).

D2.B.1 General

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability, and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD implementation of AS 4100. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D2.B.1.1 Strength Limit States

Strength design capacities ($\phi Ru$) are calculated and compared to user-defined design action effects ($S^*$), so as to ensure that $S^* \leq \phi Ru$ in accordance with AS 4100 3.4. Details for design capacity calculations are outlined in the sections that follow.

D2.B.1.2 Deflection Limit States

STAAD.Pro’s AS 4100 implementation does not generally check deflections. It is left to the user to check that both local member and frame deflections are within acceptable limits.

Note: Local member deflections parallel to the local member y-axis can be checked against a user-defined maximum “span / deflection” ratio. This can be performed using the DFF, DJ1, and DJ2 design parameters, however this is only available for MEMBER Design. Details are provided in the sections that follow.

D2.B.1.3 Eccentric Beam Reactions

STAAD.Pro does not automatically account for minimum eccentricity distances for beam reactions being transferred to columns as per AS 4100 4.3.4. However member offsets can be used to model these eccentricities.

Refer to TR.25 Member Offset Specification (on page 2499) for further information on the Member Offset feature.
D2.B.1.4 Limit States Not Considered

The following limit states are not directly considered in STAAD.Pro's implementation of AS 4100.

Table 121: Limit States Not Considered in STAAD.Pro AS 4100 Design

<table>
<thead>
<tr>
<th>Limit State</th>
<th>Code Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stability</td>
<td>AS 4100 3.3</td>
</tr>
<tr>
<td>Serviceability</td>
<td>AS 4100 3.5</td>
</tr>
<tr>
<td>Brittle Fracture</td>
<td>AS 4100 3.7</td>
</tr>
<tr>
<td>Fire</td>
<td>AS 4100 3.9</td>
</tr>
<tr>
<td>Other Design Requirements</td>
<td>AS 4100 3.11</td>
</tr>
</tbody>
</table>

D2.B.1.5 Connection Design

STAAD.Pro and Bentley's RAM Connection program currently do not support design of connections in accordance with AS 4100. In some cases connection design may govern the size of members. Such considerations are not considered in STAAD.Pro's AS 4100 and should be checked by separately.

D2.B.1.6 Bolts and Welds

Bolt holes and welds are not generally considered in STAAD.Pro's AS 4100 member design.

Note: NSC and NSF design parameters (on page 1641) are used to manually specify a reduction in net section area for compression or tension capacity calculations. These can be used to account for bolt hole area reductions. Further details are provided in the sections that follow.

D2.B.2 Analysis Methodology

Either the elastic or dynamic analysis methods may be used to obtain the forces and moments for design as per AS 4100 section 4.4. Analysis is done for the specified primary and repeat loading conditions. Therefore, it is your responsibility to enter all necessary loads and load combination factors for design in accordance with the AS/NZS 1170 Series or other relevant design codes. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

Note: Plastic analysis and design in accordance with AS 4100 section 4.5 is not implemented in STAAD.Pro.

D2.B.2.1 Elastic Analysis

Two types of elastic analysis can be performed using STAAD.Pro in accordance with AS 4100:

i. First Order Linear, Elastic Analysis - used to perform a regular elastic stiffness analysis as per AS 4100 4.4.2.1. Refer to TR.37.1 Linear Elastic Analysis (on page 2796) for additional details on this feature.

ii. Second Order PDelta Linear, Elastic Analysis - Depending on the type of structure, a PDelta analysis may be required in order to capture second-order effects as per AS 4100 4.4.1.2. Second-order effects can be
captured in STAAD.Pro by performing a PDelta second-order elastic analysis as per AS 4100 Appendix E.
Refer to TR.37.2 P-Delta Analysis Options (on page 2797) for additional details on this feature.

**Note:** Moment amplification as per AS 4100 clause 4.4.2 is not considered.

**Tip:** In order to correctly capture second-order effects for combination load cases using a PDelta Analysis, the Repeat Load feature must be used. Second-order effects will not be correctly evaluated if the Load Combination feature is used. Load Combinations are combinations of results where Repeat Loads instruct the program to perform the analysis on the combined load actions. Refer to TR.32.11 Repeat Load Specification (on page 2770) for additional details on using Repeat Loads.

### D2.B.2.2 Dynamic Analysis

Dynamic analysis may also be performed and the results combined with static analysis results. Refer TR.32.10 Dynamic Loading Specification (on page 2686) for further information on Dynamic Loading and Analysis features.

### D2.B.3 Member Property Specifications

For specification of member properties, either the steel section library available in STAAD or the User Table facility may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. For more information on these facilities, refer to G.6 Member Properties (on page 2322).

### D2.B.4 Built-in Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members. An example of the member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

#### Table 122: Available Australian Sections for STAAD.Pro AS 4100 Design

<table>
<thead>
<tr>
<th>General Profile Type</th>
<th>Australian Sections</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>I-SECTION</td>
<td>WB, WC</td>
<td>Welded beams and columns</td>
</tr>
<tr>
<td></td>
<td>UB, UC</td>
<td>Universal beams and columns</td>
</tr>
<tr>
<td>T-SECTION</td>
<td>BT, CT</td>
<td>Tees cut from universal beams and columns</td>
</tr>
<tr>
<td>CHANNEL</td>
<td>PFC</td>
<td>Parallel flange channels</td>
</tr>
<tr>
<td>ANGLE</td>
<td>EA, UA</td>
<td>Equal and unequal angles</td>
</tr>
<tr>
<td>TUBE</td>
<td>SHS, RHS</td>
<td>Square and rectangular hollow sections</td>
</tr>
</tbody>
</table>
### General Profile Type | Australian Sections | Description
---|---|---
PIPE | CHS | Circular hollow sections

**Note:** STAAD.Pro will not design the following section types to AS 4100: Double Profiles (D), Composite Sections (C), Top Cover Plates (TC), Bottom Cover Plates (BC), and Top & Bottom Cover Plates (TB), Double Channels (D, BA, & FR) and Double Angles (LD & SD). Refer to Section Profile Tables in the Graphical Environment for these options.

**Tip:** When adding and assigning sections using the built-in steel section library through the Graphical Environment, STAAD.Pro’s default tables are American. To change the default tables to Australian, select **File > Configuration** from the STAAD.Pro Start page (no input file open). Set the Default Profile Table to Australian on the Configure Program dialog Section Profile Table.

Following are the descriptions of different types of sections.

Refer to [G.6.2 Built-In Steel Section Libraries](on page 2325) for additional information.

**D2.B.4.1 UB Shapes**
These shapes are designated in the following way.

| 20 TO 30 TA ST UB150X14.0 |
| 36 TO 46 TA ST UB180X16.1 |

**D2.B.4.2 UC Shapes**
The designation for the UC shapes is similar to that for the UB shapes.

| 25 TO 35 TA ST UC100X14.8 |
| 23 56 TA ST UC310X96.8 |

**D2.B.4.3 Welded Beams**
Welded Beams are designated in the following way.

| 25 TO 35 TA ST WB700X115 |
| 23 56 TA ST WB1200X455 |

**D2.B.4.4 Welded Columns**
Welded Columns are designated in the following way.

| 25 TO 35 TA ST WC400X114 |
| 23 56 TA ST WC400X303 |

**D2.B.4.5 Parallel Flange Channels**
Shown below is the syntax for assigning names of channel sections.

| 1 TO 5 TA ST PFC75 |
| 6 TO 10 TA ST PFC380 |
### D2.B.4.6 Double Channels

Back-to-back double channels, with or without a spacing between them, are available. The letter D in front of the section name will specify a double channel.

<table>
<thead>
<tr>
<th>Member</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>TA D PFC230</td>
<td>back-to-back double channel PFC230 with no spacing in between.</td>
</tr>
<tr>
<td>17</td>
<td>TA D C230X75X25 SP 0.5</td>
<td>double channel PFC300 with a spacing of 0.5 length units between the channels.</td>
</tr>
</tbody>
</table>

In the above set of commands, member 11 is a back-to-back double channel PFC230 with no spacing in between. Member 17 is a double channel PFC300 with a spacing of 0.5 length units between the channels.

### D2.B.4.7 Angles

Two types of specification may be used to describe an angle. The standard angle section is specified as follows:

<table>
<thead>
<tr>
<th>Member</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>TA ST A30X30X6</td>
<td>angle with legs of length 30 mm and a leg thickness of 6 mm.</td>
</tr>
</tbody>
</table>

The above section signifies an angle with legs of length 30 mm and a leg thickness of 6 mm. This specification may be used when the local Z axis corresponds to the z-z axis specified in Chapter 2. If the local Y axis corresponds to the z-z axis, type specification "RA" (reverse angle) may be used.

<table>
<thead>
<tr>
<th>Member</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>TA RA A150X150X16</td>
<td>reverse angle with legs of length 150 mm and a leg thickness of 16 mm.</td>
</tr>
</tbody>
</table>

**Note:** Single angles must be specified with an “RA” (Single Angle w/Reverse Y-Z Axis) in order to be designed to AS 4100. This is to ensure that the major and minor principal axes align with the local member z and y axes respectively, similar to other section profiles.

### D2.B.4.8 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by means of input of the words SD or LD, respectively, in front of the angle size. In case of an equal angle, either SD or LD will serve the purpose.

<table>
<thead>
<tr>
<th>Member</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>33</td>
<td>TA SD A65X50X5 SP 0.6</td>
<td>short leg back-to-back double angle</td>
</tr>
<tr>
<td>37</td>
<td>TA LD A75X50X6</td>
<td>long leg back-to-back double angle</td>
</tr>
<tr>
<td>43</td>
<td>TA LD A100X75X10 SP 0.75</td>
<td>long leg back-to-back double angle</td>
</tr>
</tbody>
</table>

### D2.B.4.9 Tubes (Rectangular or Square Hollow Sections)

Tubes can be assigned in 2 ways. In the first method, the designation for the tube is as shown below. This method is meant for tubes whose property name is available in the steel table. In these examples, members 1 to 5 consist of a 2X2X0.5 inch size tube section, and members 6 to 10 consist of 10X5X0.1875 inch size tube section. The name is obtained as 10 times the depth, 10 times the width, and 16 times the thickness.

<table>
<thead>
<tr>
<th>Member</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TO 5 TA ST TUB20202.5</td>
<td>2X2X0.5 inch section</td>
</tr>
<tr>
<td>6</td>
<td>TO 10 TA ST TUB100503.0</td>
<td>10X5X0.1875 inch section</td>
</tr>
</tbody>
</table>

In the second method, tubes are specified by their dimensions. For example,

<table>
<thead>
<tr>
<th>Member</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>TA ST TUBE DT 8.0 WT 6.0 TH 0.5</td>
<td>a tube that has a height of 8 length units, width of 6 length units, and a wall thickness of 0.5 length units.</td>
</tr>
</tbody>
</table>

### D2.B.4.10 Pipes (Circular Hollow Sections)

Pipes can be assigned in 2 ways. In the first method, the designation for the pipe is as shown below. This method is meant for pipes whose property name is available in the steel table.

<table>
<thead>
<tr>
<th>Member</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TO 5 TA ST PIP180X5</td>
<td>180X5 inch section</td>
</tr>
<tr>
<td>6</td>
<td>TO 10 TA ST PIP273X6.5</td>
<td>273X6.5 inch section</td>
</tr>
</tbody>
</table>
In the second method, pipe sections may be provided by specifying the word PIPE followed by the outside and inside diameters of the section. For example,

```
1 TO 9 TA ST PIPE OD 25.0 ID 20.0
```

specifies a pipe with outside diameter of 25 length units and inside diameter of 20 length units. Only code checking, no member selection, will be performed on pipes specified in this latter manner.

### D2.B.4.11 Sample File Containing Australian Shapes

```
STAAD SPACE
UNIT METER KN
JOINT COORD
1 0 0 0 11 100 0 0
MEMB INCI
1 1 2 10
UNIT CM
MEMBER PROPERTIES AUSTRALIAN
* UB SHAPES
1 TA ST UB200X25.4
* UC SHAPES
2 TA ST UC250X89.5
* CHANNELS
3 TA ST PFC125
* DOUBLE CHANNELS
4 TA D PFC200
* ANGLES
5 TA ST A30X30X6
* REVERSE ANGLES
6 TA RA A150X150X16
* DOUBLE ANGLES - SHORT LEGS BACK TO BACK
7 TA SD A65X50X5 SP 0.6
* DOUBLE ANGLES - LONG LEGS BACK TO BACK
8 TA LD A100X75X10 SP 0.75
* TUBES (RECTANGULAR OR SQUARE HOLLOW SECTIONS)
9 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
* PIPES (CIRCULAR HOLLOW SECTIONS)
10 TA ST PIPE OD 25.0 ID 20.0
PRINT MEMB PROP
FINISH
```
b. user-defined materials

Refer TR.26.2 Specifying Constants for Members and Elements (on page 2503) for further information on the Built-in Material Constants feature.

Refer TR.26.1 Define Material (on page 2501) for further information on the Define Material feature.

D2.B.6.1 Young’s Modulus of Elasticity (E)

STAAD.Pro’s default steel material’s E value is 205,000 MPa. However AS 4100 section 1.4 states that the modulus of elasticity should be taken as 200,000 MPa. There are a number of options to change this value:

- change the steel material through the input file or GUI for each file created
- define a new steel material for each file created
- change the default STAAD.Pro metric E value in the file C:/Windows/StaadPro20070.ini, going to the “[Material-Metric]” section, and changing \E1=205.0e6\ to \E1=200.0e6\. Restart STAAD.Pro for this to take effect.

Caution: Virtualization features of Windows 7, Windows 8, and Windows 10 may require additional files to be modified. Contact Bentley Technical Support for assistance.

D2.B.7 Member Resistances

The member resistance is calculated in STAAD according to the procedures outlined in AS 4100. Calculated design capacities are compared to corresponding axial, bending moment, and shear forces determined from the STAAD.Pro analysis. These are used to report the fail or pass status for the members designed.

Two types of design checks are typically performed per AS 4100:

- Nominal section checks
- Nominal member checks

The nominal section capacity refers to the capacity of a cross-section to resists applied loads, and accounts for cross-section yielding and local buckling effects. The nominal member capacity on the other hand refers to the capacity of a member to resist applied loads, and includes checks for global member buckling effects including Euler buckling, lateral-torsional buckling, etc.

D2.B.7.1 Axial Tension

The criteria governing the capacity of tension members are based on two limit states per AS 4100 Section 7. The limit state of yielding of the gross section is intended to prevent excessive elongation of the member.

The second limit state involves fracture at the section with the minimum effective net area \(\phi N_t\) section axial tension capacities are calculated (Cl.7.2). Through the use of the NSF parameter (see D2.B.8 Design Parameters (on page 1641)), you may specify the net section area. STAAD calculates the tension capacity of a member based on these two limit states per Cl.7.1 and Cl.7.2 respectively of AS 4100. Eccentric end connections can be taken into account using the KT correction factor, per Cl.7.3. The \(f_y\) yield stress is based on the minimum plate yield stress. Parameters FYLD, FU, and NSF are applicable for these calculations.

D2.B.7.2 Axial Compression

The compressive strength of members is based on limit states per AS 4100 Section 6. It is taken as the lesser of nominal section capacity and nominal member capacity. Nominal section capacity, \(\phi N_s\), is a function of form factor (Cl.6.2.2), net area of the cross section, and yield stress of the material. Through the use of the NSC
parameter (see D2.B.8 Design Parameters (on page 1641)), you may specify the net section area. Note that this parameter is different from that corresponding to tension. The program automatically calculates the form factor. The \( k_f \) form factors are calculated based on effective plate widths per Cl.6.2.4, and the \( f_y \) yield stress is based on the minimum plate yield stress.

Nominal member capacity, \( \varphi N_c \), is a function of nominal section capacity and member slenderness reduction factor (Cl.6.3.3). This value is calculated about both principal x and y axes. Here, you are required to supply the value of \( \alpha_b \) (Cl.6.3.3) through the ALB parameter (see D2.B.8 Design Parameters (on page 1641)). The effective length for the calculation of compressive strength may be provided through the use of the parameters \( K_Y \), \( K_Z \), LY, and LZ (see D2.B.8 Design Parameters (on page 1641)).

### D2.B.7.3 Bending

Bending capacities are calculated to AS 4100 Section 5. The allowable bending moment of members is determined as the lesser of nominal section capacity and nominal member capacity (ref. Cl.5.1).

The nominal section moment capacity, \( \varphi M_s \), is calculated about both principal x and y axes and is the capacity to resist cross-section yielding or local buckling and is expressed as the product of the yield stress of the material and the effective section modulus (ref. Cl.5.2). The effective section modulus is a function of section type (on page 1638) (i.e., compact, noncompact, or slender) and minimum plate yield stress \( f_y \). The nominal member capacity depends on overall flexural-torsional buckling of the member (ref.Cl.5.3).

**Note:** For sections where the web and flange yield stresses (\( f_{y,\text{web}} \) and \( f_{y,\text{flange}} \) respectively) are different, the lower of the two yield stresses is applied to both the web and flange to determine the slenderness of these elements.

Member moment capacity, \( \varphi M_b \), is calculated about the principal x axis only (ref. Cl.5.6). Critical flange effective cross-section restraints and corresponding design segment and sub-segments are used as the basis for calculating capacities.

### D2.B.7.4 Interaction of Axial Force and Bending

Combined section bending and shear capacities are calculated using the shear and bending interaction method as per Cl.5.12.3.

**Note:** This check is only carried out where \( \varphi V \) section web shear capacities are calculated. Refer Table 1B.6-1 for details.

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. Here, the adequacy of a member is also examined against both section (ref. Cl.8.3.4) and member capacity (ref.Cl.8.4.5). These account for both in-plane and out-of-plane failures. If the summation of the left hand side of the equations, addressed by the above clauses, exceeds 1.0 or the allowable value provided using the RATIO parameter (see D2.B.8 Design Parameters (on page 1641)), the member is considered to have FAILED under the loading condition.

### D2.B.7.5 Shear

Section web shear capacity, \( \varphi V \), is calculated per Cl.5.11, including both shear yield and shear buckling capacities. Once the capacity is obtained, the ratio of the shear force acting on the cross section to the shear capacity of the section is calculated. If any of the ratios (for both local Y & Z-axes) exceed 1.0 or the allowable value provided using the RATIO parameter (see D2.B.8 Design Parameters (on page 1641)), the section is considered to have failed under shear.

Table 1B.6-1 below highlights which shear capacities are calculated for different profile types.
Table 123: Section Type Shear Checks

<table>
<thead>
<tr>
<th>General Profile Type</th>
<th>Australian Section</th>
<th>Shear Checks</th>
</tr>
</thead>
<tbody>
<tr>
<td>I-SECTION (i.e., parallel to minor principal y-axis)</td>
<td>WB, WC, UB, UC</td>
<td>Calculated for web only</td>
</tr>
<tr>
<td>T-SECTION</td>
<td>BT, CT</td>
<td></td>
</tr>
<tr>
<td>CHANNEL</td>
<td>PFC</td>
<td></td>
</tr>
<tr>
<td>ANGLE</td>
<td>EA, UA</td>
<td>No checks performed</td>
</tr>
<tr>
<td>TUBE</td>
<td>SHS, RHS</td>
<td>Calculated parallel to both x &amp; y principal axes</td>
</tr>
<tr>
<td>PIPE</td>
<td>CHS</td>
<td>Per AS 4100 5.11.4</td>
</tr>
</tbody>
</table>

**Note:** Only unstiffened web capacities are calculated. Stiffened webs are not considered. Bearing capacities are not considered.

**D2.B.7.6 Torsion**

STAAD.Pro does not design sections or members for torsion for AS 4100.

**D2.B.8 Design Parameters**

The design parameters outlined in Table 1B.1 are used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application’s specific needs. The design scope indicates whether design parameters are applicable for MEMBER Design, PMEMBER Design, or both.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Table 124: Australian Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Design Scope</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>MEMBER, PMEMBER</td>
<td>Must be specified as AUSTRALIAN to invoke design per AS 4100 - 1998. Design code to follow. See [TR.48.1 Parameter Specifications](on page 2851).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Design Scope</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| ALB            | 2.0           |             | Member section constant (refer cl. 6.3.3)  
If ALB is 2.0, it is automatically calculated based on TABLE 6.3.3(1), 6.3.3(2); otherwise the input value is used. |
| ALM            | 0.0           |             | Moment modification factor (refer cl. 5.6.1.1)  
If ALM is 0.0, it is automatically calculated based cl. 5.6.1.1; otherwise the input value is used. |
| BEAM           | 0.0           |             | 0.0 = design only for end moments and those at locations specified by SECTION command.  
1.0 = Perform design for moments at twelfth points along the beam. |
| DFF            | None         | Analytical members only | "Deflection Length" / Maximum Allowable local deflection. |
| DJ1            | Start Joint of member | Analytical members only | Joint No. denoting start point for calculation of "deflection length" |
| DJ2            | End Joint of member | Analytical members only | Joint No. denoting end point for calculation of "deflection length" |
| DMAX           | 45.0 [in.]    |             | Maximum allowable depth (Applicable for member selection) |
| DMIN           | 0.0 [in.]     |             | Minimum required depth (Applicable for member selection) |
| FU             | 500.0 [MPa]   |             | Ultimate strength of steel. |
| FYLD           | 250.0 [MPa]   |             | Yield strength of steel. |
| IST            | 1             |             | Steel type - 1 - SR, 2 - HR, 3 - CF, 4 - LW, 5 - HW  
| KT             | 1.0           |             | Correction factor for distribution of forces (refer cl. 7.2) |
| KY             | 1.0           |             | K value for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio. |
| KZ             | 1.0           |             | K value for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio. |
### Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Design Scope</th>
<th>Description</th>
</tr>
</thead>
</table>
| LHT            | 0             | Physical members only | Load height position as described in Table 5.6.3(2) of AS 4100:1998  
0 = at Shear center  
1 = At top flange |
| LY             | Member Length |             | Length for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio. |
| LZ             | Member Length |             | Length for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio. |
| MAIN           | 0.0           |             | A value of either 0.0 or 1.0 suppresses the slenderness ratio check. Checks are not explicitly required per AS 4100.  
Any value greater than 1.0 is used as the limit for slenderness in compression. |
| NSC            | 1.0           |             | Net section factor for compression members = An / Ag  
(refer cl. 6.2.1) |
| NSF            | 1.0           |             | Net section factor for tension members. |
| PBRACE         | None          | Physical members only | Refer to section 1B.11 (on page 1649) for details on the PBRACE parameter. |
| PHI            | 0.9           |             | Capacity reduction factor |
| RATIO          | 1.0           |             | Permissible ratio of actual load effect to the design strength. |
### Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Design Scope</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGR</td>
<td>0</td>
<td></td>
<td>Steel Grade. Refer to Note a below.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0 = Default (see note below)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1.0 = high strength grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2 = AS/NZS 3679.1 350</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3 = AS/NZS 3679.1 300</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>4 = AS/NZS 1163 C450</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>5 = AS/NZS 1163 C350</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>6 = AS/NZS 1163 C250</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>7 = AS/NZS 3678 450</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>8 = AS/NZS 3678 400</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>9 = AS/NZS 3678 350</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>10 = AS/NZS 3678 WR350</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>11 = AS/NZS 3678 300</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>12 = AS 3597 500</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>13 = AS 3597 600</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>14 = AS 3597 700</td>
</tr>
<tr>
<td>SKL</td>
<td>1.0</td>
<td></td>
<td>A load height factor given in Table 5.6.3(2)</td>
</tr>
<tr>
<td>SKR</td>
<td>1.0</td>
<td></td>
<td>A lateral rotation restraint factor given in Table 5.6.3(3)</td>
</tr>
<tr>
<td>SKT</td>
<td>1.0</td>
<td></td>
<td>A twist restraint factor given in Table 5.6.3(1)</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td></td>
<td>Output detail</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.0 = report only minimum design results</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1.0 = report design strengths in addition to TRACK output</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2.0 = provide full details of design</td>
</tr>
<tr>
<td>UNB</td>
<td>Member Length</td>
<td></td>
<td>Unsupported length in bending compression of the bottom flange for calculating moment resistance.</td>
</tr>
<tr>
<td>UNT</td>
<td>Member Length</td>
<td></td>
<td>Unsupported length in bending compression of the top flange for calculating moment resistance.</td>
</tr>
</tbody>
</table>

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### D2.B.8.1 Notes

a. DFF, DJ1, and DJ2 – Deflection calculations

Compute \[ \text{Delta} = \sqrt{((DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2)} \]
Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

**Note:** Deflection calculations are not applicable to PMEMBERs.

**a.** A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.

![Diagram of a beam with joints and deflections](image)

D = Maximum local deflection for members 1, 2, and 3.

**PARAMETERS**

DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL

**b.** If DJ1 and DJ2 are not used, “Deflection Length” will default to the member length and local deflections will be measured from original member line.

**c.** It is important to note that unless a DFF value is specified, STAAD.Pro will not perform a deflection check. This is in accordance with the fact that there is no default value for DFF.

**b.** LHT Parameter

If the shear force is constant within the segment, longitudinal position of the load is assumed to be at the segment end.

If there is any variation of the shear force and the load is acting downward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU) Kl is assumed to be 1.4.

If there is any variation of the shear force and the load is acting upward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU) Kl is assumed to be 1.0 as the load acting at the top flange is contributing to stabilize against local torsional buckling.

**c.** SGR Parameter

Typically SGR 2-3 should be used for rolled profiles, SRG 4-6 should be used for hollow profiles and SRG 7-11 should be used for welded sections. When the default value (0) is used, the steel grade for different types of sections is selected by the program as follows:

**Table 125: Steel Grades used for the SGR Parameter**

<table>
<thead>
<tr>
<th>Section Type</th>
<th>Steel Grade Used</th>
</tr>
</thead>
<tbody>
<tr>
<td>WB, WC, Tee section cut from WB and WC, other welded and UPT sections</td>
<td>AS 3678 300</td>
</tr>
</tbody>
</table>
### Section Type

<table>
<thead>
<tr>
<th>Section Type</th>
<th>Steel Grade Used</th>
</tr>
</thead>
<tbody>
<tr>
<td>UB, UC, Tee section cut from UB and UC, EA, UA and all UPT sections</td>
<td>AS 3679.1 300</td>
</tr>
<tr>
<td>UB, UC, Tee section cut from UB and UC, EA, UA and all other rolled sections</td>
<td></td>
</tr>
<tr>
<td>Pipe, Tube, CHS, RHS, SHS Pipe, Tube, CHS, RHS, SHS</td>
<td>AS 1163 C250</td>
</tr>
</tbody>
</table>

**Note:** If a value for the FYLD parameter has been specified, then that value will be used. Otherwise, the SGR value will be used to determine the yield strength and tensile strength values for the steel, based on maximum thickness of the individual elements of the section. Only for shear capacity calculation web thickness is used. Similarly, Tensile Strength is determined either from FU parameter or from SGR parameter.

**Caution:** A check is introduced to see if yield stress is more than 450 MPa or not. If it is, a warning is issued and the yield stress is set to 450 MPa.

### D2.B.8.2 Example

The following example uses the member design facility in STAAD.Pro. However, it is strongly recommended to use the Physical member design capabilities for AS 4100:

```plaintext
PARAMETER 1
CODE AUSTRALIAN
ALB 0.0 MEMBER ALL
ALM 1.13 MEMBER ALL
BEAM 1.0 MEMBER ALL
DFF 250.0 MEMBER ALL
DMAX 0.4 MEMBER ALL
DMIN 0.25 MEMBER ALL
FU 400.0 MEMBER ALL
FYLD 310.0 MEMBER ALL
IST 2.0 MEMBER ALL
KT 0.85 MEMBER ALL
KX 0.75 MEMBER ALL
KY 1.0 MEMBER ALL
LX 4.5 MEMBER ALL
LY 6.0 MEMBER ALL
MAIN 1.0 MEMBER ALL
NSC 0.9 MEMBER ALL
NSF 1.0 MEMBER ALL
PHI 0.9 MEMBER ALL
RATIO 0.9 MEMBER ALL
SGR 1.0 MEMBER ALL
SKT 1.0 MEMBER ALL
SKL 1.0 MEMBER ALL
SKR 1.0 MEMBER ALL
TRACK 2.0 MEMBER ALL
UNB 3.4 MEMBER ALL
UNT 6.8 MEMBER ALL
CHECK CODE MEMBER ALL
```
D2.B.9 Code Checking

The purpose of code checking is to evaluate whether the provided section properties of the members are adequate for the specified loads as per AS 4100 requirements.

**Tip:** The member selection (on page 1648) facility can be used to instruct the program to select a different section if the specified section is found to be inadequate.

Code checking for an analytical member is done using forces and moments at every twelfth point along the beam. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. The extent of detail of the output can be controlled by using the TRACK parameter.

**Note:** Code checking cannot be performed on composite and prismatic sections.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

Example of commands for code checking:

```plaintext
UNIT NEWTON METER
PARAMETER
CODE AUSTRALIAN
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
CHECK CODE MEMB 3 4
```

D2.B.9.1 Physical Members

For physical members (PMEMBERs), code checks are performed at section stations positioned at 1/12th points along each analytical member included in the PMEMBER. It is up to you to determine if these locations cover critical sections for design, and adjust as necessary. The number of stations for PMEMBER Design cannot be altered, however the analytical members can be split so that in effect more stations are checked for a PMEMBER.

For each section station along a PMEMBER, section capacity checks are carried for design actions at that station location. Member capacity checks are also carried out for each station. For these the program searches each side of the station to find adjacent effective restraints and design forces and moments. This allows the program to determine the segment / sub-segment that the section station resides in, and then proceeds to calculate the member capacities. Enough section stations should be included to capture all segments / sub-segments for checking.

**Note:** When checking combined actions for the section capacities, the design actions at the section station are used. However when checking combined actions for the member capacities, the maximum forces from anywhere along the segment / sub-segment being considered are used. This is as stipulated in AS 4100 8.2.

The output reports whether the member has PASSed or FAILed the design checks, as well as the critical condition, critical load case, magnitudes of design actions for the most critical cross-section location (distance from the start joint), and complete calculations for design. The TRACK design parameter can be used to control the level of detail provided in the output.
**Tip:** You can use the **Utilization Ratio** tool in the graphical user interface **Postprocessing** workflow to view color-coded results.

In some cases some of the output will report “N/A” values. This occurs where a calculation does not apply to a member. For example if a member never goes into tension then no values can be reported in the tension capacity output sections.

**Note:** As per AS 4100 1.4, the TRACK 2.0 detailed level of output for PMEMBER Design uses x and y subscripts to refer to major and minor principal axes respectively. These differ to STAAD.Pro local member axes, where z and y refer to major and minor principal axes.

### D2.B.10 Member Selection

This process incrementally checks increasing section profile sizes until a size is found that is AS 4100 compliant, or the largest section has been checked. Only section profiles of the same type as modeled are incrementally checked, with the increasing sizes based on a least weight per unit length criteria.

For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table.

The design calculations for Member Selection are the same as for **Code Checking** (on page 1647).

**Tip:** A Fixed Group command is also available, and can be used to force all members within a user-defined group to take the same section size based on the most critical governing design criteria for all members within that group. This is particularly useful when you want to use the Member Selection feature, but want a group of elements to have the same size. Refer to Section 5.49 of the Technical Reference Manual for information on using this feature.

**Note:** Member Selection will change member sizes, and hence will change the structure's stiffness matrix. In order to correctly account for this, a subsequent analysis and Code Check should be performed to ensure that the final structure is acceptable. This may need to be carried out over several iterations.

**Note:** Composite and prismatic sections cannot be selected.

Refer to **D1.B.1.4 Member Selection** (on page 1419) for general information on Member Selection. Refer to **TR.49.1 Member Selection Specification** (on page 2853) for details the specification of the Member Selection command.

```
Example of commands for member selection:

UNIT NEWTON METER
PARAMETER
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
SELECT MEMB 3 4
```
D2.B.11 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. The term CRITICAL COND refers to the section of the AS 4100 specification which governs the design.

D2.B.12 Physical Member Design

There are two methods available in STAAD.Pro for checking members against the requirements of AS 4100:

a. Analytical member method
b. Physical member method

Herein these are referred to as MEMBER Design and PMEMBER Design respectively.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 2) build 2007.07 or higher.

Traditionally STAAD.Pro performed code checks based on single analytical members (i.e., single members between two nodes). This implementation remains in place as shown in the example in Section 1B.8 (on page 1641). Physical Member (PMEMBER) Design on the other hand allows you to group single or multiple analytical members into a single physical design member for the purposes of design to AS 4100.

PMEMBER Design also has additional features, including:

- automated steel grades based on section type;
- automated tensile stress \( f_u \) and yield stress \( f_y \) values based on plate thicknesses;
- automated segment / sub-segment design;
- improved detailed design calculation output; and

Thus, it is strongly recommended that PMEMBER Design be used, even for the design of single analytical members.

D2.B.12.1 Modeling with Physical Members

Physical Members may be grouped by either of the following methods:

- STAAD.Pro Editor - Directly specify physical members in the input file. Refer to TR.16.2 Physical Members (on page 2443) for additional information.
- Graphical Environment - Using the tools in the Physical Member group on the Geometry ribbon tab, members can be manually or automatically formed.

**Note:** When creating PMEMBERs for AS 4100, this must be performed in STAAD.Pro’s M. Creating Model Objects (on page 639). Do not use the Steel Design workflow.

D2.B.12.2 Segment and Sub-Segment Layout

For calculation of member bending capacities about the principal x-axis, the PMEMBER Design uses the concept of segment / sub-segment design. By default PMEMBERs are automatically broken up into design segments and sub-segments based on calculated effective restraints. User-defined restraints assigned using the PBRACE design parameter are checked to see if they are effective (i.e., if they are placed on the critical flange as per AS 4100 5.5). Restraints not applied to the critical flange are ineffective and hence are completely ignored.

Refer to Section 1B.7 for further information on how user-defined restraints are applied using the PBRACE design parameter, including available restraint types, and restraint layout rules.
Note: Segment and sub-segment layouts for PMEMBERs may change for different load cases considered for design. Some restraints may be effective for one particular load case as they are found to apply to the critical flange, however for another load case may be found not to act on the critical flange, and found to be ineffective. In other words the critical flange can change for each load case considered.

Typically the critical flange will be the compression flange, except for segments with a "U" restraint at one end, in which case it will be the tension flange (as is the case for a cantilever).

The PMEMBER Design uses the following routine to determine effective cross-section restraints for each load case considered:

i. first all user-defined restraints are checked to see if they are applied to the compression flange, with those that aren't ignored;

ii. next a check is made to see if a “U” type restraint is found at either end of the PMEMBER. If this is the case then any adjacent “L” restraints up to the next “F”, “FR”, “P” or “PR” restraint are also ignored, regardless of whether they are placed on the critical or non-critical flange. Refer AS 4100 5.4.2.4.

The compression flange in step 1 of the routine above is calculated based on the bending moments at the locations of the restraints being considered. If the bending moment is zero at the same location as a restraint then the following method is used to determine which flange is critical at the zero moment location:

a. If the zero moment is at the end of the PMEMBER, then the compression flange is based on the bending moment at a small increment from then end;

b. If the zero moment is along the PMEMBER and is a peak value, then the compression flange is based on the bending moment at a small increment from that location;

c. If neither 1 or 2 above is valid, then the stiffer of the restraints at that location is taken. The stiffness of different restraint types from the most stiff to least stiff are taken as outlined in Table 1B.9-3.

Table 126: Assumed Order of Restraint Stiffness for Zero Moment Critical Flange

<table>
<thead>
<tr>
<th>Stiffness</th>
<th>Restraint Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Most Stiff</td>
<td>FR</td>
</tr>
<tr>
<td>↓</td>
<td>F</td>
</tr>
<tr>
<td>↓</td>
<td>PR</td>
</tr>
<tr>
<td>↓</td>
<td>P</td>
</tr>
<tr>
<td>↓</td>
<td>L</td>
</tr>
<tr>
<td>↓</td>
<td>U</td>
</tr>
<tr>
<td>Least Stiff</td>
<td>None</td>
</tr>
</tbody>
</table>

Once the effective restraints have been determined, the PMEMBER is divided into segments bounded by “F”, “P”, “FR”, “PR” or “U” effective restraints. These segments are then further divided into sub-segments by effective “L” restraints.

Note: Sub-segment lengths are not automatically checked to determine if they provide full lateral restraint as per AS 4100 5.3.2.4.
For design of cantilevers, the free tip should have user-defined “U” restraints applied to both top and bottom flanges.

**Note:** If the effective restraints for any load case consist of “U” or “L” restraints only, an error will be reported.

### D2.B.12.3 Automated PMEMBER Design Calculations

The AS 4100 PMEMBER Design automates many design calculations, including those required for segment / sub-segment design.

#### Table 127: Automated PMEMBER AS 4100 Design Parameters and Calculations

<table>
<thead>
<tr>
<th>Automated Design Calculations</th>
<th>PMEMBER Design Parameter</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha_b$ compression member section constant per AS 4100 6.3.3.</td>
<td>ALB</td>
<td></td>
</tr>
<tr>
<td>$\alpha_m$ moment modification factor per AS 4100 5.6.1.1.</td>
<td>ALM</td>
<td>Calculated based on moments distribution for individual segments and sub-segments.</td>
</tr>
<tr>
<td>$f_u$ tensile strength per AS 4100 2.1.2.</td>
<td>FU</td>
<td>Based on nominal steel grade specified using SGR design parameter and section type.</td>
</tr>
<tr>
<td>$f_y$ yield stress per AS 4100 2.1.1.</td>
<td>FYLD</td>
<td>Based on nominal steel grade specified using SGR design parameter and section type.</td>
</tr>
<tr>
<td>residual stress category for AS 4100 Table 5.2 and AS 4100 Table 6.2.4.</td>
<td>IST</td>
<td>Based on section type.</td>
</tr>
<tr>
<td>correction factor for distribution of forces in a tension member per AS 4100 7.3.</td>
<td>KT</td>
<td>Based on section type and eccentric end connection specified using EEC design parameter.</td>
</tr>
<tr>
<td>Load height position for automated calculation of the $k_l$ load height factor per AS 4100 Table 5.6.3(2).</td>
<td>LHT</td>
<td>LHT is used for automating calculation of $k_l$ load height factors for segments and sub-segments, per AS 4100 Table 5.6.3(2). See “Load Height Position” for details.</td>
</tr>
<tr>
<td>Segment and sub-segment layout.</td>
<td>PBRACE</td>
<td>Refer to the Segment and Sub-Segment Layout section above for details.</td>
</tr>
<tr>
<td>Nominal steel grade.</td>
<td>SGR</td>
<td>Based on section types.</td>
</tr>
</tbody>
</table>
### Automated Design Calculations

<table>
<thead>
<tr>
<th>Automated Design Calculations</th>
<th>PMEMBER Design Parameter</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k_t$ twist restraint factor as per AS 4100 Table 5.6.3(1).</td>
<td>SKT</td>
<td>Based on effective end restraints for each segment / sub-segment.</td>
</tr>
<tr>
<td>$k_l$ load height factor as per AS 4100 Table 5.6.3(2).</td>
<td>SKL</td>
<td>Based on effective end restraints for each segment / sub-segment, and LHT design parameter (refer above).</td>
</tr>
<tr>
<td>$k_r$ lateral rotation restraint factor as per AS 4100 Table 5.6.3(3).</td>
<td>SKR</td>
<td>Based on effective end restraints for each segment / sub-segment. This is where the distinction between &quot;F&quot; and &quot;FR&quot;, as well as “P&quot; and “PR&quot; is used.</td>
</tr>
</tbody>
</table>

### D2.B.12.4 Load Height Position

When LHT is set to 1.0 to specify a top flange load height position, STAAD.Pro takes the top to be the positive local y-axis of the member.

**Note:** This may not literally be the top flange for say a column or beam with a beta angle. The local member axes can be viewed in the GUI by selecting "Beam Orientation" in the Diagrams Labels dialog (or Ctrl+O keyboard shortcut).

To automate $k_l$ using AS 4100 Table 5.6.3(2), the longitudinal position of the load also needs to be considered, i.e., as either “within segment” or “at segment end”.

To determine which of these applies, the shear forces at the ends of each design segment / sub-segment is considered. If the shear force is found to have the same direction and magnitude at both ends, it is assumed that loads act at the segment end.

If on the other hand the shear force at each end is found to have different directions or magnitudes, loads are assumed to act within the segment.

**Note:** The above method includes an allowance for the self-weight of the member to be considered, as the self-weight always acts through the shear center.

The net sum of the end shears is also used to determine if the load is acting in the positive or negative local member y-axis direction. If LHT is set to 1.0 for top flange loading, the net sum is used to determine whether the top flange loading is acting to stabilise or destabilise the member for lateral torsional buckling. Negative local y-axis net loads act to destabilise the segments / sub-segments, whereas positive local y-axis net loads act to stabilise segments / sub-segments.

### D2.B.12.5 Example

```
PARAMETER 1
CODE AUSTRALIAN
DMAX 0.4 PMEMBER 1 TO 42
DMIN 0.25 PMEMBER 1 TO 42
KX 0.75 PMEMBER 1 TO 42
KY 1.0 PMEMBER 1 TO 42
LX 4.5 PMEMBER 1 TO 42
LY 6.0 PMEMBER 1 TO 42
```
**D2.B.12.6 Physical Member Restraints Specification**

The `PBRACE` parameter is used to specify the restraint condition along the top and bottom flange of a `PMEMBER`.

**General Format**

```
PBRACE { TOP | BOTTOM } f1 r1 f2 r2 ... f52 r52 (PMEMBER pmember-list)
```

- **fn** a fraction of the `PMEMBER` length where restraint condition is being specified. This value is any ratio between 0.0 and 1.0.
- **rn** one of the possible restraint conditions as follows:

**Table 128: Physical Member Restraint Types**

<table>
<thead>
<tr>
<th>Designation, rn</th>
<th>Restraint Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>Fully restrained</td>
<td></td>
</tr>
<tr>
<td>P</td>
<td>Partially restrained</td>
<td></td>
</tr>
<tr>
<td>L</td>
<td>Laterally restrained</td>
<td>Cannot be specified at the ends of design members.</td>
</tr>
<tr>
<td>U</td>
<td>Unrestrained</td>
<td>Can only be applied at the ends of design members, and must be applied to both flanges to be effective.</td>
</tr>
<tr>
<td></td>
<td><strong>Caution:</strong> Both top and bottom flanges cannot be unrestrained at the same location (as this is unstable).</td>
<td></td>
</tr>
<tr>
<td>FR</td>
<td>Fully and rotationally restrained</td>
<td></td>
</tr>
<tr>
<td>PR</td>
<td>Partially and rotationally restrained</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>Continuously restrained</td>
<td>The flange is assumed to be continuously supported at that flange up to next restraint location. For continuously supported flange unbraced length is assumed to be zero.</td>
</tr>
</tbody>
</table>
Description

Refer to AS 4100 Section 5.5 for a full definition of the critical flange. Typically this will be the compression flange, except for segments with U restraint at one end, then it will be the tension flange (as is the case for cantilever portion at the end).

- when gravity loads are dominant (i.e., negative local y-axis direction), the critical flange of a segment shall be the top flange (i.e., tension).
- when upward wind loads are dominant (i.e., positive local y-axis direction), the critical flange shall be the bottom flange (i.e., tension).

Design physical members are divided into segments by "F", "P", "FR", "PR" or "U" effective section restraints. Segments are further broken down into sub-segments by "L" restraints, but only if the "L" restraints are deemed to be "effective". "L" restraints are only considered to be effective when positioned on the "critical" flange between "F", "P", "FR" or "FP" restraints. If an "L" restraint is positioned on the non-critical flange it shall be completely ignored. Further, if an "L" restraint is positioned between a "U" and an "F", "P", "FR" or "PR" restraint, it shall be ignored (regardless of whether it is on the critical or non-critical flange).

Design members must have either a F, P, FR, PR, or U restraint specified at both ends, for both flanges.

- If UNL is not specified, segment length is used as UNL and used as L in effective length calculation as per 5.6.3.
- If ALM i.e., α_m is not provided, automatic calculation of ALM is done based on moments within the segment.
- If SKR i.e., Kr is not provided, it is automatically calculated based on table 5.6.3(3) considering restraint conditions are the end of the segment. If FR or PR is found at only one of the end, Kr is assumed to be 0.85; if FR or PR is found at both the ends, 0.70 is used as Kr.
- If SKT i.e., Kt is not provided, it is automatically calculated based on Table 5.6.3(1) considering end restraints of the segment and section geometric information and segment length.
- If SKL i.e., Kl is not provided, it is automatically calculated based on Table 5.6.3(2) considering end restraints of the segment, Load Height Position parameter, LHT and shear force variation within the segment.

Notes

a. If PMEMBER list is not provided, all the PMEMBERS are restrained by same configuration.
b. It is not necessary to provide the restraint locations in sequence as the program sorts them automatically.
c. Unless specified, PMEMBER ends are assumed to be Fully Restrained (F).
d. While designing any section of the member, effective restraints are searched on each side of the section along the critical flange.
e. The types of restraints applied to the top and bottom flanges at each location determines the effective section restraints. These are outlined in the table below:

Table 129: Restraint Meanings in Critical and Noncritical Flanges

<table>
<thead>
<tr>
<th>Case</th>
<th>Flange</th>
<th>Restraint on a Critical Flange</th>
<th>Restraint on a Non-Critical Flange</th>
<th>Effective Section Restraint</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>U</td>
<td>U</td>
<td>U</td>
<td>U</td>
</tr>
</tbody>
</table>
## Design

### D. Design Codes

<table>
<thead>
<tr>
<th>Case</th>
<th>Flange</th>
<th>Restraint on a Critical Flange</th>
<th>Restraint on a Non-Critical Flange</th>
<th>Effective Section Restraint</th>
</tr>
</thead>
<tbody>
<tr>
<td>II</td>
<td>1</td>
<td>L</td>
<td>Nothing</td>
<td>L</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Nothing</td>
<td>L</td>
<td>None</td>
</tr>
<tr>
<td>III</td>
<td>1</td>
<td>P or F</td>
<td>Nothing or U</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Nothing or U</td>
<td>P or F</td>
<td>P</td>
</tr>
<tr>
<td>IV</td>
<td>1</td>
<td>PR or FR</td>
<td>Nothing or U</td>
<td>FR</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Nothing or U</td>
<td>PR or FR</td>
<td>PR</td>
</tr>
<tr>
<td>V</td>
<td>1</td>
<td>L, P or F</td>
<td>L, P, F, FR or PR</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>FR or PR</td>
<td>L, P, F, FR or PR</td>
<td>FR</td>
</tr>
</tbody>
</table>

**Note:** The critical flange can change for each load case considered.

f. If a C-type restraint is defined to a flange at a particular location of the PMEMBER, program will consider that continuous restraint to be effective up to the next bracing point of the point of contraflexure whichever is nearer to the continuous restraint.

**Related Links**

- M. To manually add physical member restraints (on page 652)

### D3. British Codes

#### D3.A. British Codes - Concrete Design per BS8110

**Note:** This code has been removed from the batch design. To perform design to the current BS 8110 design code, please use the interactive design in the D. British Concrete design per BS 8110 (on page 1081).

#### D3.B. British Codes - Steel Design per BS5950:2000

STAAD.Pro is capable of performing steel design based on the British code BS 5950-1:2000 *Structural use of steelwork in building - Part 1: Code of practice for design - Rolled and welded sections*, Incorporating Corrigendum No. 1.

#### D3.B.1 General

The design philosophy embodied in BS5950:2000 is built around the concept of limit state design, used today in most modern steel design codes. Structures are designed and proportioned taking into consideration the limit states at which they become unfit for their intended use. Two major categories of limit state are recognized - serviceability and ultimate. The primary considerations in ultimate limit state design are strength and stability.
while that in serviceability limit state is deflection. Appropriate safety factors are used so that the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of BS5950:2000, members are proportioned to resist the design loads without exceeding the limit states of strength and stability. Accordingly, the most economic section is selected on the basis of the least weight criteria. This procedure is controlled by the designer in specification of allowable member depths, desired section type or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The complete B.S.C. steel tables for both hot rolled and hollow sections are built into the program for use in specifying member properties as well as for the actual design process. See section 2B.4 for information regarding the referencing of these sections. In addition to universal beams, columns, joists, piles, channels, tees, composite sections, beams with cover plates, pipes, tubes, and angles, there is a provision for user provided tables.

STAAD.Pro 2006 and later have the additional option to design tapered I shaped (wide flange) beams according to Annex G of BS5950. See Section 2B.13 (on page 1675) for a complete description.

**Single Angle Sections**

Angle sections are un-symmetrical and when using BS 5950:2000 table 25 you must consider four axes: two principal, u-u and v-v and two geometric, a-a and b-b. The effective length for the v-v axis, L vv, is taken as the $L_{VV} \cdot K_Y$ if not specified. The a-a and b-b axes are determined by which leg of the angle is fixed by the connection and should be specified using the LEG parameter, see section 5B.6 for more information on the LEG parameter. The effective length in the a-a axis is taken as $L_{XY} \cdot K_Y$ and the effective length in the b-b axis as $L_{LZ} \cdot K_Z$.

The following diagram shows the axes for angles which have been defined with either an ST or RA specification and is connected by its longer leg (i.e., a-a axis is parallel to the longer leg).
D3.B.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D3.B.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. Any user-defined section may be specified, except for GENERAL or PRISMATIC sections. For more information on these facilities, refer to G.6 Member Properties (on page 2322).

D3.B.4 Built-In Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Almost all BSI steel sections are available for input. A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Following are the descriptions of different types of sections available:

Refer to G.6.2 Built-In Steel Section Libraries (on page 2325) for additional information.

D3.B.4.1 Universal Beams, Columns, and Piles

All rolled universal beams, columns and pile sections are available. The following examples illustrate the designation scheme.

```
20 TO 30 TA ST UB305X165X54
33 36 TA ST UC356X406X287
100 102 106 TA ST UP305X305X186
```

D3.B.4.2 Rolled Steel Joists

Joist sections may be specified as they are listed in BSI-80 with the weight omitted. In those cases where two joists have the same specifications but different weights, the lighter section should be specified with an "A" at the end.

```
10 TO 20 TA ST J0152X127
1 2 TA ST J0127X114A
```

D3.B.4.3 Channels

All rolled steel channel sections from the BSI table have been incorporated in STAAD. The designation is similar to that of the joists. The same designation scheme as in BSI tables may be used with the weight omitted.

```
10 TO 15 TA ST CH305X102
55 57 59 61 TA ST CH178X76
```
D3.B.4.4 Double Channels

Back-to-back double channels, with or without spacing between them, are available. The letter "D" in front of the section name will specify a double channel (e.g., D CH102X51, D CH203X89, etc.)

51 52 53 TA D CH152X89
70 TO 80 TA D CH305X102 SP 5.

(specifies a double channel with a spacing of 5 length units)

Note: Face-to-face double channels cannot be used in a CHECK CODE command.

D3.B.4.5 Tee Sections

Tee sections are not input by their actual designations, but instead by referring to the universal beam shapes from which they are cut. For example,

54 55 56 TA T UB254X102X22

(tee cut from UB254X102X22)

D3.B.4.6 Angles

All equal and unequal angles are available for analysis. Two types of specifications may be used to describe an angle section, either a standard, ST specification or reversed angle, RA specification. Note, however, that only angles specified with an RA specification can be designed.

The standard angle section is specified as follows:

15 20 25 TA ST UA200X150X18

This specification may be used when the local STAAD z-axis corresponds to the V-V axis specified in the steel tables. If the local STAAD y-axis corresponds to the V-V axis in the tables, type specification "RA" (reverse angle) may be used.

35 TO 45 TA RA UA200X150X18

D3.B.4.7 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle, either LD or SD will serve the purpose. For example,

14 TO 20 TA LD UA200X200X16 SP 1.5
23 27 TA SD UA80X60X6

"SP" denotes spacing between the individual angle sections.

Note: If the section is defined from a Double Angle User Table, then the section properties must be defined with an 11th value which defines the radius of gyration about an individual sections’ principal v-v axis (See Technical Reference Manual, 5.19 User Steel Table Specification)

D3.B.4.8 Pipes (Circular Hollow Sections)

To designate circular hollow sections from BSI tables, use PIP followed by the numerical value of diameter and thickness of the section in mm omitting the decimal section of the value provided for diameter. The following example will illustrate the designation.

10 15 TA ST PIP213.2
(specifies a 21.3 mm dia. pipe with 3.2 mm wall thickness)

Circular hollow sections may also be provided by specifying the outside and inside diameters of the section. For example,

```
1 TO 9 TA ST PIPE OD 25.0 ID 20.0
```

(specifies a pipe with outside dia. of 25 and inside dia. of 20 in current length units)

Only code checking and no member selection will be performed if this type of specification is used.

**D3.B.4.9 Rectangular or Square Hollow Sections (Tubes)**

Designation of tubes from the BSI steel table is illustrated below:

```
Figure 174: BSI tube nomenclature
```

Example:

```
15 TO 25 TA ST TUB160808.0
```

Tubes, like pipes, can also be input by their dimensions (Height, Width and Thickness) and not by any table designations.

```
6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```

(a tube that has a height of 8, a width of 6, and a wall thickness of 0.5 length units)

**Note:** Only code checking and no member selection is performed for TUBE sections specified this way.

**D3.B.5 Member Capacities**

The basic measure of capacity of a beam is taken as the plastic moment of the section. This is a significant departure from the standard practice followed in BS449, in which the limiting condition was attainment of yield stress at the extreme fibers of a given section. With the introduction of the plastic moment as the basic measure of capacity, careful consideration must be given to the influence of local buckling on moment capacity. To assist this, sections are classified as either Class 1, plastic, Class 2, compact, Class 3, semi-compact or Class 4, slender, which governs the decision whether to use the plastic or the elastic moment capacity. The section classification is a function of the geometric properties of the section. STAAD is capable of determining the section classification for both hot rolled and built up sections. In addition, for slender sections, BS5950 recommends the use of a 'stress reduction factor' to reduce the design strength. This factor is again a function of the geometry of the section and is automatically determined by STAAD for use in the design process.

**D3.B.5.1 Axial Tension**

In members with axial tension, the tensile load must not exceed the tension capacity of the member. The tension capacity of the member is calculated on the basis of the effective area as outlined in Section 4.6 of the code.
STAAD calculates the tension capacity of a given member per this procedure, based on a user supplied net section factor (NSF—a default value of 1.0 is present but may be altered by changing the input value—see D3.B.6 Design Parameters (on page 1661)), proceeding with member selection or code check accordingly. BS5950 does not have any slenderness limitations for tension members.

**D3.B.5.2 Compression**

Compression members must be designed so that the compression resistance of the member is greater than the axial compressive load. Compression resistance is determined according to the compressive strength, which is a function of the slenderness of the gross section, the appropriate design strength and the relevant strut characteristics. Strut characteristics take into account the considerable influence residual rolling and welding stresses have on column behavior. Based on data collected from extensive research, it has been determined that sections such as tubes with low residual stresses and Universal Beams and Columns are of intermediate performance. It has been found that I-shaped sections are less sensitive to imperfections when constrained to fail about an axis parallel to the flanges. These research observations are incorporated in BS5950 through the use of four strut curves together with a selection of tables to indicate which curve to use for a particular case. Compression strength for a particular section is calculated in STAAD according to the procedure outlined in Annex C of BS5950 where compression strength is seen to be a function of the appropriate Robertson constant (representing Strut Curve) corresponding Perry factor, limiting slenderness of the member and appropriate design strength.

A departure from BS5950:1990, generally compression members are no longer required to be checked for slenderness limitations, however, this option can be included by specifying a MAIN parameter. Note, a slenderness limit of 50 is still applied on double angles checked as battenstruts as per clause 4.7.9.

**D3.B.5.3 Axially Loaded Members With Moments**

In the case of axially loaded members with moments, the moment capacity of the member must be calculated about both principal axes and all axial forces must be taken into account. If the section is plastic or compact, plastic moment capacities will constitute the basic moment capacities subject to an elastic limitation. The purpose of this elastic limitation is to prevent plasticity at working load. For semi-compact or slender sections, the elastic moment is used. For plastic or compact sections with high shear loads, the plastic modulus has to be reduced to accommodate the shear loads. The STAAD implementation of BS5950 incorporates the procedure outlined in section 4.2.5 and 4.2.6 to calculate the appropriate moment capacities of the section.

For members with axial tension and moment, the interaction formula as outlined in section 4.8.2 is applied based on effective tension capacity.

For members with axial compression and moment, two principal interaction formulae must be satisfied—Cross Section Capacity check (4.8.3.2) and the Member Buckling Resistance check (4.8.3.3). Three types of approach for the member buckling resistance check have been outlined in BS5950:2000—the simplified approach (4.8.3.3.1), the more exact approach (4.8.3.3.2) and Annex I1 for stocky members. As noted in the code, in cases where neither the major axis nor the minor axis moment approaches zero, the more exact approach may be more conservative than the simplified approach. It has been found, however, that this is not always the case and STAAD therefore performs both checks, comparing the results in order that the more appropriate criteria can be used.

Additionally the equivalent moment factors, $m_x$, $m_y$, and $m_{yx}$, can be specified by the user or calculated by the program.

Members subject to biaxial moments in the absence of both tensile and compressive axial forces are checked using the appropriate method described above with all axial forces set to zero. STAAD also carries out cross checks for compression only, which for compact/plastic sections may be more critical. If this is the case, COMPRESSION will be the critical condition reported despite the presence of moments.
D3.B.5.4 Shear

A member subjected to shear is considered adequate if the shear capacity of the section is greater than the shear load on the member. Shear capacity is calculated in STAAD using the procedure outlined in section 4.2.3, also 4.4.5 and Annex H3 if appropriate, considering the appropriate shear area for the section specified.

Since plastic moment capacity is the basic moment capacity used in BS5950, members are likely to experience relatively large deflections. This effect, coupled with lateral torsional buckling, may result in severe serviceability limit state. Hence, lateral torsional buckling must be considered carefully.

The procedure to check for lateral torsional buckling as outlined in section 4.3 has been incorporated in the STAAD implementation of BS5950. According to this procedure, for a member subjected to moments about the major axis, the ‘equivalent uniform moment’ on the section must be less than the lateral torsional buckling resistance moment. For calculation of the buckling resistance moment, the procedure outlined in Annex B.2 has been implemented for all sections with the exception of angles. In Annex B.2., the resistance moment is given as a function of the elastic critical moment, Perry coefficient, and limiting equivalent slenderness, which are calculated within the program; and the equivalent moment factor, $m_{LT}$, which is determined as a function of the loading configuration and the nature of the load (stabilizing, destabilizing, etc).

D3.B.5.6 RHS Sections - Additional Provisions

Rectangular Hollow sections are treated in accordance with S.C.I. recommendations in cases when the plastic axis is in the flange. In such cases, the following expressions are used to calculate the reduced plastic moduli:

For $n \geq 2t(D-2t)/A$

$$S_{rx} = \frac{A^2}{4(B-t)}\left(1 - n\right)\left[\frac{2D(B-t)}{A} + n - 1\right]$$

For $n \geq 2t(B-2t)/A$

$$S_{ry} = \frac{A^2}{4(D-t)}\left(1 - n\right)\left[\frac{2B(D-t)}{A} + n - 1\right]$$

D3.B.6 Design Parameters

Available design parameters to be used in conjunction with BS5950 are listed in table 2B.1 along with their default values.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as BS5950 Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
</tbody>
</table>

Note: Once a parameter is specified, its value stays at that specified number till it is specified again. This is the way STAAD works for all codes.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>AD</strong></td>
<td>Depth at end/2</td>
<td>Distance between the reference axis and the axis of restraint. See G. 2.3</td>
</tr>
</tbody>
</table>
| **BEAM**       | 3.0           | Beam divisions  
0. Design only for end moments or those locations specified by the SECTION command.  
1. Calculate forces and moments at 12th points along the member. Establish the location where Mz is the maximum. Use the forces and moments at that location. Clause checks at one location.  
2. Same as BEAM = 1.0 but additional checks are carried out for each end.  
3. Calculate moments at 12th points along the member. Clause checks at each location including the ends of the member. |
| **CAN**        | 0             | Deflection check method. See Note 1 below.  
0. Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2.  
1. Deflection check based on the principle that maximum deflection is of the cantilever type (see note below) |
| **CB**         | 1             | Specifies the method used to calculate Mb.  
1. Value of Mb from Clause 4.3.6 is used (default).  
2. Value of Mbs from Clause 4.7.7 is used. |
| **DFF**        | None(Mandatory for deflection check, TRACK 4.0) | "Deflection Length" / Maxm. allowable local deflection  
See Note 1d below. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length.&quot; See Note 1 below.</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length.&quot; See Note 1 below.</td>
</tr>
<tr>
<td>DMAX *</td>
<td>100.0cm</td>
<td>Maximum allowable depth</td>
</tr>
<tr>
<td>DMIN *</td>
<td>0.0 cm</td>
<td>Minimum allowable depth</td>
</tr>
<tr>
<td>ESTIFF</td>
<td>0.0</td>
<td>Clauses 4.8.3.3.1 and 4.8.3.3.2. 0.0 = Fail ratio uses MIN of 4.8.3.3.1, 4.8.3.3.2. and Annex I1 checks. 1.0 = Fail ratio uses MAX of 4.8.3.3.1, 4.8.3.3.2. and Annex I1 checks.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K factor value in local y - axis. Usually, this is the minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K factor value in local z - axis. Usually, this is the major axis.</td>
</tr>
<tr>
<td>LEG</td>
<td>0.0</td>
<td>Valid range from 0 – 7 and 10. The values correspond to table 25 of BS5950 for fastener conditions. See note 2 below.</td>
</tr>
<tr>
<td>LVV *</td>
<td>Maximum of Lyy and Lzz(Lyy is a term used by BS5950)</td>
<td>Used in conjunction with LEG for Lvv as per BS5950 table 25 for double angles. See note 6 below.</td>
</tr>
<tr>
<td>LY *</td>
<td>Member Length</td>
<td>Length in local y - axis (current units) to calculate (KY)(LY)/Ryy slenderness ratio.</td>
</tr>
<tr>
<td>LZ *</td>
<td>Member Length</td>
<td>Length in local z - axis (current units) to calculate (KZ)(LZ)/Rzz slenderness ratio.</td>
</tr>
</tbody>
</table>
### Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MLT</td>
<td>1.0</td>
<td>Equivalent moment factor for lateral torsional buckling as defined in clause 4.8.3.3.4</td>
</tr>
<tr>
<td>MX</td>
<td>1.0</td>
<td>Equivalent moment factor for major axis flexural buckling as defined in clause 4.8.3.3.4</td>
</tr>
<tr>
<td>MY</td>
<td>1.0</td>
<td>Equivalent moment factor for minor axis flexural buckling as defined in clause 4.8.3.3.4</td>
</tr>
<tr>
<td>MYX</td>
<td>1.0</td>
<td>Equivalent moment factor for minor axis lateral flexural buckling as defined in clause 4.8.3.3.4</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
</tbody>
</table>
| PNL *          | 0.0           | Transverse stiffener spacing ("a" in Annex H1)  
|                |               | 0.0 = Infinity  
|                |               | Any other value used in the calculations. |
| PY *           | Set according to steel grade (SGR) | Design strength of steel |
| MAIN           | 0.0           | Slenderness limit for members with compression forces, effective length/ radius of gyration, for a given axis:  
|                |               | 0.0 = Slenderness not performed.  
|                |               | 1.0 = Main structural member (180)  
|                |               | 2.0 = Secondary member. (250)  
<p>|                |               | 3.0 = Bracing etc (350) |
| RATIO          | 1.0           | Permissible ratio of the actual capacities. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| SAME**        | 0.0          | Controls the sections to try during a SELECT process.  
|               |              | 0.0 = Try every section of the same type as original  
|               |              | 1.0 = Try only those sections with a similar name as original, e.g., if the original is an HEA 100, then only HEA sections will be selected, even if there are HEM's in the same table. |
| SBLT          | 0.0          | Identify Section type for section classification  
|               |              | 0.0 = Rolled Section  
|               |              | 1.0 = Built up Section  
|               |              | 2.0 = Cold formed section |
| SWAY          | none         | Specifies a load case number to provide the sway loading forces in clause 4.8.3.3.4 (See additional notes) |
| SGR           | 0.0          | Steel Grade per BS4360  
|               |              | 0.0 = Grade S 275  
|               |              | 1.0 = Grade S 355  
|               |              | 2.0 = Grade S 460  
|               |              | 3.0 = As per GB 1591 – 16 Mn |
| TB            | 0.0          | Limit of moment capacity in Cl 4.2.5.1:  
|               |              | 0 = Mc limit 1.5pyZ  
|               |              | 1 = Mc limit 1.2 pyZ |
### Design

**D. Design Codes**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Output details</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0 = Suppress all member capacity info.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = Print all member capacities.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.0 = Print detailed design sheet.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4.0 = Deflection Check (separate check to main select / check code)</td>
</tr>
<tr>
<td>UNF</td>
<td>1.0</td>
<td>Factor applied to unsupported length for Lateral Torsional Buckling effective length per section 4.3.6.7 of BS5950.</td>
</tr>
<tr>
<td>UNL *</td>
<td>Member Length</td>
<td>Unsupported Length for calculating Lateral Torsional Buckling resistance moment section 4.3.6.7 of BS5950.</td>
</tr>
<tr>
<td>WELD</td>
<td>1.0 closed 2.0 open</td>
<td>Weld Type</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = Closed sections. Welding on one side only (except for webs of wide flange and tee sections)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.0 = Open sections. Welding on both sides (except pipes and tubes)</td>
</tr>
</tbody>
</table>

* current units must be considered.

**For angles, if the original section is an equal angle, then the selected section will be an equal angle and vice-versa for unequal angles.

**Note:** There was an NT parameter in STAAD.Pro 2005 build 1003 which is now automatically calculated during the design as it is load case dependent.

### D3.B.6.1 Notes

1. **CAN, DJ1, and DJ2 – Deflection**

   a. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. Refer to [TR.44 Printing Section Displacements for Members](on page 2846) for details on local displacement.
If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1, DZ1) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at DJ2 or the end node of the member.

Compute \( \Delta = \sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2} \)

Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

Then, if CAN is specified a value 1, \( dff = \frac{L}{\Delta} \)

Ratio due to deflection = \( \frac{dff}{dff} \)

b. If CAN = 0, deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.

D = Maximum local deflection for members 1, 2, and 3.

PARAMETERS

| DFF 300. ALL |
| DJ1 1 ALL |
| DJ2 4 ALL |

c. If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

d. It is important to note that unless a DFF value is specified, STAAD will not perform a deflection check. This is in accordance with the fact that there is no default value for DFF.

e. The above parameters may be used in conjunction with other available parameters for steel design.

2. LEG – follows the requirements of BS5950 table 28. This table concerns the fastener restraint conditions for angles, double angles, tee sections and channels for slenderness. The following values are available:

Table 131: LEG Parameter values

<table>
<thead>
<tr>
<th>Clause</th>
<th>Bold Configuration</th>
<th>Leg</th>
<th>LEG Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.7.10.2 Single Angle</td>
<td>(a) - 2 bolts</td>
<td>short leg</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>long leg</td>
<td>3.0</td>
</tr>
<tr>
<td></td>
<td>(b) - 1 bolts</td>
<td>short leg</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>long leg</td>
<td>2.0</td>
</tr>
</tbody>
</table>
The slenderness of single and double angle, channel and tee sections are specified in BS 5950 table 25 depending on the connection provided at the end of the member. To define the appropriate connection, a LEG parameter should be assigned to the member.

The following list indicates the value of the LEG parameter required to match the BS5950 connection definition:

Clause 4.7.10.2 Single Angle:

a. 2 Bolts: Short leg = 1.0, Long Leg = 3.0
b. 1 Bolt: Short Leg = 0.0, Long Leg = 2.0

For single angles, the slenderness is calculated for the geometric axes, a-a and b-b as well as the weak v-v axis. The effective lengths of the geometric axes are defined as:

\[ L_a = K_Y \times L_Y \]
\[ L_b = K_Z \times L_Z \]

The slenderness calculated for the v-v axis is then used to calculate the compression strength \( p_c \) for the weaker principal axis (z-z for ST angles or y-y for RA specified angles). The maximum slenderness of the a-a and b-b axes is used to calculate the compression strength \( p_c \) for the stronger principal axis.

Alternatively for single angles where the connection is not known or Table 25 is not appropriate, by setting the LEG parameter to 10, slenderness is calculated for the two principal axes y-y and z-z only. The LVV parameter is not used.

For double angles, the LVV parameter is available to comply with note 5 in table 25. In addition, if using double angles from user tables, (Technical Reference Manual section 5.19) an eleventh value, \( r_{vv} \) should be
supplied at the end of the ten existing values corresponding to the radius of gyration of the single angle making up the pair.

3. **PY – Steel Design Strength**

   The design parameter PY should only be used when a uniform design strength for an entire structure or a portion thereof is required. Otherwise the value of PY will be set according to the stipulations of BS5950 table 9 in which the design strength is seen as a function of cross sectional thickness for a particular steel grade (SGR parameter) and particular element considered. Generally speaking this option is not required and the program should be allowed to ascertain the appropriate value.

4. **UNL, LY, and LZ – Relevant Effective Length**

   The values supplied for UNL, LY and LZ should be real numbers greater than zero in current units of length. They are supplied along with or instead of UNF, KY and KZ (which are factors, not lengths) to define lateral torsional buckling and compression effective lengths respectively. Please note that both UNL or UNF and LY or KY values are required even though they are often the same values. The former relates to compression flange restraint for lateral torsional buckling while the latter is the unrestrained buckling length for compression checks.

5. **TRACK – Control of Output Formats**

   When the TRACK parameter is set to 0.0, 1.0, or 2.0, member capacities will be printed in design related output (code check or member selection) in kilonewtons per square meter.

   TRACK 4.0 causes the design to carry out a deflection check, usually with a different load list to the main code check. The members that are to be checked must have the parameters DFF, DJ1, and DJ2 set.

6. **MX, MY, MYX, and MLT – Equivalent Moment Factors**

   The values for the equivalent moment factors can either be specified directly by the user as a positive value between 0.4 and 1.0 for MX, MY and MYX and 0.44 and 1.0 for MLT.

   The program can be used to calculate the values for the equivalent moment factors by defining the design member with a GROUP command (see the Technical Reference Manual section 5.16 Listing of Members/Elements/Joints by Specification of GROUPS). The nodes along the beam can then be defined as the location of restraint points with J settings.

   Additionally for the MLT parameter, the joint can be defined as having the upper flange restrained (positive local Y) with the a U setting or the lower flange restrained (negative local Y) with a L setting.

   For example, consider a series of 5 beam elements as a single continuous member as shown below:
To enable the steel design, the beam needs to be defined as a group, called MainBeam:

```
START  GROUP DEFINITION
MEMBER
_MainBeam  11 2 38 12 3
END  GROUP DEFINITION
```

Note: This can be done in the User Interface by selecting Tools > Create New Group....

Therefore, this 5 beam member has 6 joints such that:

- Joint 1 = Node 3
- Joint 2 = Node 1
- Joint 3 = Node 33
- Joint 4 = Node 14
- Joint 5 = Node 7
- Joint 6 = Node 2

a. Consider MX, MY and MYX

Say that this member has been restrained in its’ major axis (local Y) only at the ends. In the minor axis (local Z) it has been restrained at the ends and also at node number 33 (joint 3). For local flexural buckling, it has only been restrained at its ends. Hence:

For the major axis, local Y axis:

MX _MainBeam J1 J6

For the minor axis, local Z axis:

MY _MainBeam J1 J3 J6

For the lateral flexural buckling, local X axis:

MYX _MainBeam J1 J6

b. Consider MLT
Say that this member has been restrained at its' ends against lateral torsional buckling and the top flange has been restrained at node number 33 (joint 3) and only the lower flange at node number 7, (joint 5).
Hence:

MLT _MainBeam J1 T3 L5 J6

To split the beam into two buckling lengths for $L_y$ at joint 14:

MY _groupname J1 J4 J6

7. SWAY – Sway Loadcase

This parameter is used to specify a load case that is to be treated as a sway load case in the context of clause 4.8.3.3.4. This load case would be set up to represent the $k_{am}pM_k$ mentioned in this clause and the steel design module would add the forces from this load case to the forces of the other load case it is designed for.

Note that the load case specified with this parameter will not be designed as a separate load case. The following is the correct syntax for the parameter:

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SWAY</td>
<td>(load case number)</td>
<td>ALL MEMBER (member list)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(group name)</td>
</tr>
</tbody>
</table>

Example

**SWAY 5 MEM 1 to 10**

**SWAY 6 _MainBeams**

D3.B.7 Design Operations

STAAD.Pro contains a broad set of facilities for the design of structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem.

The operations to perform a design are:

- Specify the load cases to be considered in the design; the default is all load cases.
- Specify design parameter values, if different from the default values.
- Specify whether to perform code checking or member selection along with the list of members.

These operations may be repeated by the user any number of times depending upon the design requirements.

D3.B.8 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate. The adequacy is checked as per BS5950. Code checking is done using the forces and moments at specific sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks; the critical condition of BS5950 code (like any of the BS5950 specifications for compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other
specified RATIO value); the governing load case, and the location (distance from the start of the member of forces in the member where the critical condition occurs).

Code checking can be done with any type of steel section listed in Section 2B.4 (on page 1657) or any of the user defined sections as described in G.6.3 User-Provided Steel Table (on page 2325), except profiles defined in GENERAL and ISECTION tables.

**Note:** PRISMATIC sections are also not acceptable steel sections for design per BS5950 in STAAD.Pro.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

### D3.B.9 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN, which limits the maximum and minimum depth of the members.

Member selection can be performed with all the types of steel sections with the same limitations as defined in section 2B.8 Code Checking (on page 1671).

Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table.

Member selection cannot be performed on members whose section properties are input as prismatic or as above limitations for code checking.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

### D3.B.10 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

- **MEMBER** refers to the member number for which the design is performed.
- **TABLE** refers to steel section name, which has been checked against the steel code or has been selected.
- **RESULTS** prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
- **CRITICAL COND** refers to the section of the BS5950 code which governs the design.
- **RATIO** prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
- **LOADING** provides the load case number, which governed the design
- **FX, MY, and MZ** provide the axial force, moment in local Y-axis and the moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.
LOCATION specifies the actual distance from the start of the member to the section where design forces govern.

TRACK
If the parameter TRACK is set to 1.0, the program will block out part of the table and will print the allowable bending capacities in compression (MCY & MCZ) and reduced moment capacities (MRY & MRZ), allowable axial capacity in compression (PC) and tension (PT) and shear capacity (PV). TRACK 2.0 will produce the design results as shown in section 2B.9 (on page 1672).

An example of each TRACK setting follows:

**D3.B.10.1 Example output for TRACK 0.0**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST UC305X305X118 PASS</td>
<td>BS-4.3.6</td>
<td>LOCATION</td>
</tr>
<tr>
<td></td>
<td>179.66 C</td>
<td>0.769</td>
</tr>
</tbody>
</table>

**D3.B.10.2 Example output for TRACK 1.0**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST UC305X305X118 PASS</td>
<td>BS-4.3.6</td>
<td>LOCATION</td>
</tr>
<tr>
<td></td>
<td>179.66 C</td>
<td>0.769</td>
</tr>
</tbody>
</table>

**MATERIAL DATA**
Grade of steel = S 275
Modulus of elasticity = 210 kN/mm2
Design Strength (py) = 265 N/mm2

**SECTION PROPERTIES (units - cm)**
Member Length = 600.00
Gross Area = 150.00 Net Area = 127.50 Eff. Area = 150.00

Moment of inertia : z-z axis 27700.004
Plastic modulus : 1671.526
Elastic modulus : 589.460
**Effective modulus** : 1960.000 895.000  
**Shear Area** : 103.471 37.740  

**DESIGN DATA (units - kN,m)** BS5950-1/2000  
**Section Class** : PLASTIC  
**Squash Load** : 3975.00  
**Axial force/Squash load** : 0.045  

<table>
<thead>
<tr>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression Capacity : 351.7</td>
<td>2455.9</td>
</tr>
<tr>
<td>Moment Capacity : 519.4</td>
<td>234.3</td>
</tr>
<tr>
<td>Reduced Moment Capacity : 516.9</td>
<td>234.3</td>
</tr>
<tr>
<td>Shear Capacity : 1645.2</td>
<td>600.1</td>
</tr>
</tbody>
</table>

**BUCKLING CALCULATIONS (units - kN,m)**  
(axis nomenclature as per design code)  

<table>
<thead>
<tr>
<th>x-x axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness : 44.153</td>
<td>77.203</td>
</tr>
<tr>
<td>Radius of gyration (cm) : 13.589</td>
<td>7.772</td>
</tr>
<tr>
<td>Effective Length : 6.000</td>
<td>6.000</td>
</tr>
</tbody>
</table>

LTB Moment Capacity (kNm) and LTB Length (m): 435.00, 6.000  
LTB Coefficients & Associated Moments (kNm):  
mlt = 1.00 : mx = 1.00 : my = 1.00 : myx = 1.00  
Mlt = 334.46 : Mx = 334.46 : My = 0.00 : My = 0.00  

**CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):**  

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>BS-4.2.3-(Y)</td>
<td>0.143</td>
<td>3</td>
<td>-</td>
<td>85.6</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>BS-4.3.6</td>
<td>0.769</td>
<td>3</td>
<td>-</td>
<td>85.6</td>
<td>-</td>
<td>334.5</td>
<td>-</td>
</tr>
<tr>
<td>BS-4.7</td>
<td>0.098</td>
<td>1</td>
<td>239.7</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>BS-4.8.3.2</td>
<td>0.647</td>
<td>3</td>
<td>179.7</td>
<td>85.6</td>
<td>0.0</td>
<td>334.5</td>
<td>0.0</td>
</tr>
<tr>
<td>BS-4.8.3.3.1</td>
<td>0.842</td>
<td>3</td>
<td>179.7</td>
<td>-</td>
<td>-</td>
<td>334.5</td>
<td>0.0</td>
</tr>
<tr>
<td>BS-4.8.3.3.2</td>
<td>0.842</td>
<td>3</td>
<td>179.7</td>
<td>-</td>
<td>-</td>
<td>334.5</td>
<td>0.0</td>
</tr>
<tr>
<td>ANNEX I.1</td>
<td>0.714</td>
<td>3</td>
<td>179.7</td>
<td>-</td>
<td>-</td>
<td>334.5</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.

**D3.B.11 Plate Girders**

Sections will be considered for the Plate Girder checks (BS 5950 Section 4.4) if \( d/t > 70 \varepsilon \) for ‘rolled sections’ or \( d/t>62 \varepsilon \) for ‘welded sections’. The parameter SBLT should be used to identify sections as rolled or welded; see the parameter list for more information.

If the plate girder has intermediate stiffeners, the spacing is set with the PNL parameter. These are then used to check against the code clauses ‘4.4.3.2 - Minimum web thickness for serviceability’ and ‘4.4.3.3 - Minimum web thickness to avoid compression flange buckling’. The following printout is then included if a TRACK 2.0 output is selected:

Shear Buckling check is required: \( V_b = 1070 \text{ kN} : q_w = 118 \text{ N/mm}^2 \)  
\( d = 900 \text{ mm} : t = 10 \text{ mm} : a = 200 \text{ mm} : p_y = 275 \text{ N/mm}^2 \)  
BS-4.4.3.2 status = PASS : BS-4.4.3.3 status = PASS

The section is then checked for shear buckling resistance using clause “4.4.5.2 - Simplified method” and the result is included in the ratio checks.
D3.B.12 Composite Sections

Sections that have been defined as acting compositely with a concrete flange either from a standard database section using the CM option, or from a modified user WIDE FLANGE database with the additional composite parameters, cannot be designed with BS5950:2000.

D3.B.13 Design of Tapered Beams

Sections will be checked as tapered members provided that are defined either as a Tapered I section or from a USER table.

Example using a Tapered I section:

UNIT CM
MEMBER PROPERTY
1 TO 5 TAPERED 100 2.5 75 25 4 25 4

Example using a USER table:

START USER TABLE
TABLE 1
UNIT CM
ISECTION
1000mm_TAPER 100 2.5 75 25 4 25 4 0 0 0
750mm_TAPER 75 2.5 50 25 4 25 4 0 0 0
END

You must specify the effective length of unrestrained compression flange using the parameter UNL.

The program compares the resistance of members with the applied load effects, in accordance with BS 5950-1:2000. Code checking is carried out for locations specified by the user via the SECTION command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. The user may choose the degree of detail in the output data by setting the TRACK parameter.

The beam is designed as other wide flange beams apart from the Lateral Torsional Buckling check which is replaced by the Annex G.2.2. check.

D3.B.13.1 Design Equations

A beam defined with tapered properties as defined above will be checked as a regular wide flange (e.g., UB or UC), except that the following is used in place of clause 4.3.6, the lateral torsional buckling check.

D3.B.13.2 Check Moment for Taper Members as per clause G.2.2

The following criterion is checked at each defined check position in the length of the member defined by the BEAM parameter.

\[ M_{xi} \leq M_{bi} (1 - F_c/P_c) \]

where

- \( F_c \) = the longitudinal compression at the check location
- \( M_{bi} \) = the buckling resistance moment \( M_b \) from 4.3.6 for an equivalent slenderness \( \lambda_{TB} \), see G.2.4.2, based on the appropriate modulus \( S, S_{eff}, Z \) or \( Z_{eff} \) of the cross-section at the point \( i \) considered
- \( M_{xi} \) = the moment about the major axis acting at the point \( i \) considered
\[ P_c = \text{the compression resistance from 4.7.4 for a slenderness } \lambda_{TC-y}, \text{ see G.2.3, based on the properties of the minimum depth of cross-section within the segment length } L \]

**D3.B.13.2 G.2.3 Slenderness ITC**

\[ \lambda_{TC} = y \lambda \]

where

\[ y = \left[ \frac{1 + \left( \frac{2a}{h_s} \right)^2}{1 + \left( \frac{2a}{h_s} \right)^2 + 0.05 \left( \frac{\lambda}{x} \right)^2} \right]^{0.5} \]

\[ \lambda = \frac{L_y}{r_y} \]

\[ a = \text{the distance between the reference axis and the axis of restraint,} \]
\[ h_s = \text{the distance between the shear centers of the flanges} \]
\[ L_y = \text{the length of the segment} \]
\[ r_y = \text{the radius of gyration for buckling about the minor axis} \]
\[ x = \text{is the torsional index} \]

**D3.B.13.4 G.2.4.2 Equivalent slenderness ITB for tapered members**

\[ \lambda_{TB} = c n_v \nu_t \lambda \]

Where, for a two-flange haunch:

\[ \nu_t = \left[ \frac{4a}{h_s} \right]^{0.5} \]

where

\[ C = \text{the taper factor, see G.2.5} \]

**D3.B.13.5 G.2.5 Taper factor**

For an I-section with \( D \geq 1.2B \) and \( x \geq 20 \), the taper factor, \( c \), is as follows:

\[ c = 1 + \frac{3}{x - 9} \left( \frac{D_{max}}{D_{min}} - 1 \right)^{2/3} \]

where

\[ D_{max} = \text{the maximum depth of cross-section within the length } L_y, \text{ see Figure G.3} \]
\[ D_{min} = \text{the minimum depth of cross-section within the length } L_y, \text{ see Figure G.3} \]
\[ x = \text{the torsional index of the minimum depth cross-section, see 4.3.6.8} \]

Otherwise, \( c \) is taken as 1.0 (unity).

**D3.C. British Codes - Design per BS5400**


**D3.C.1 General Comments**

The British Standard, BS5400 adopts the limit state design philosophy and is applicable to steel, concrete, and composite construction. The code is in ten parts covering various aspects of bridge design. The implementation
of part 3, Code of practice for design of steel bridges, in STAAD is restricted in its scope to simply supported spans. It is assumed that the depth remains constant and both construction and composite stages of steel I-Sections can be checked. The following sections describe in more detail features of the design process currently available in STAAD.

D3.C.2 Shape Limitations

The capacity of sections could be limited by local buckling if the ratio of flange outstand to thickness is large. In order to prevent this, the code sets limits to the ratio as per clause 9.3.2. In the event of exceeding these limits, the design process will terminate with reference to the clause.

D3.C.3 Section Class

Sections are further defined as compact or noncompact. In the case of compact sections, the full plastic moment capacity can be attained. In the case of noncompact sections, local buckling of elements may occur prior to reaching the full moment capacity and for this reason the extreme fibre stresses are limited to first yield. In STAAD, section types are determined as per clause 9.3.7 and the checks that follow will relate to the type of section considered.

D3.C.4 Moment Capacity

Lateral torsional buckling may occur if a member has unrestrained elements in compression. The code deals with this effect by limiting the compressive stress to a value depending on the slenderness parameter which is a modified form of the ratio $L_e/R_y$. $L_e$ is the effective length governed by the provision of lateral restraints satisfying the requirements of clause 9.12.1. Once the allowable compressive stress is determined then the moment capacity appropriate to the section type can be calculated. STAAD takes the effective length as that provided by the user, defaulting to the length of the member during construction stage and as zero, assuming full restraint throughout, for the composite stage. The program then proceeds to calculate the allowable compressive stress based on appendix G7 from which the moment capacity is then determined.

D3.C.5 Shear Capacity

The shear capacity, as outlined in clause is a function of the limiting shear strength, $l$, which is dependant on the slenderness ratio. STAAD follows the iterative procedure of appendix G8 to determine the limiting shear strength of the web panel. The shear capacity is then calculated based on the formula given under clause 9.9.2.2.

D3.C.6 Design Parameters

Available design parameters to be used in conjunction with BS5400 are listed in table 2C.1. Depending on the value assigned to the WET parameter, you can determine the stage under consideration. For a composite design check, taking into consideration the construction stage, two separate analyses are required. In the first, member properties are non-composite and the WET parameter is set to 1.0. In the second, member properties should be changed to composite and the WET parameter set to 2.0. Member properties for composite or non-composite sections should be specified from user provided tables (refer to section 5.19 of the manual for specification of user tables). Rolled sections, composite or non-composite, come under WIDE FLANGE section-type and built-up sections under ISECTION. When specifying composite properties the first parameter is assigned a negative value and four additional parameters provided giving details of the concrete section. See user table examples provided.
Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 132: BS5400 Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ESTIFF</td>
<td>0</td>
<td>Specify the criteria used for the design of compression members with moments.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Member passes if <em>either</em> Cl. 4.8.3.3.1 <em>or</em> Cl. 4.8.3.3.2 check.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Member passes if <em>both</em> Cl. 4.8.3.3.1 <em>and</em> Cl. 4.8.3.3.2 check.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value for bending about Y-axis. Usually this is minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value for bending about Z-axis. Usually this is major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for bending about Y-axis, in current units of length.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio for bending about Z-axis, in current units of length.</td>
</tr>
<tr>
<td>MAIN</td>
<td>1.0</td>
<td>Grade of concrete:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. 30 N/mm²</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. 40 N/mm²</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. 50 N/mm²</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>PY</td>
<td>*</td>
<td>Yield stress of steel. Set according to Design Strength of steel SGR</td>
</tr>
<tr>
<td>RATIO</td>
<td>1</td>
<td>Permissible ratio of actual to allowable stresses.</td>
</tr>
</tbody>
</table>
### Parameter Name | Default Value | Description
---|---|---
SBLT | 0.0 | Steel
| | 0.0 = Rolled Section
| | 1.0 = Built up Section
SGR | 0.0 | Steel Grade per BS4360
| | 0. Grade 43
| | 1. Grade 50
| | 2. Grade 55
TRACK | 1.0 | Used to control the level of detail in the output
| | 0. Suppress all member capacities
| | 1. Print all member capacities
UNL | Member Length | Unsupported length for calculating allowable compressive bending stress, in current units of length.
WET | 0.0 | Used to specify the stage of construction.
| | 0. Wet stage with no data saved for composite stage
| | 1. Wet stage with data saved for composite stage
| | 2. Composite and wet stage combined
| | 3. Composite stage only

**D3.C.7 Composite Sections**

The definition of composite sections has been provided for in the standard sections definition (refer to TR.20.1 Assigning Properties from Steel Tables on page 2461) for details. This is purely for analysis and for obtaining the right section properties. It uses the American requirement of 18 times depth (CT) as the effective depth. For more control with British sections two new options are available in user provided tables.

**D3.C.7.1 Wide Flange Composite**

Using the standard definition of I sections in WIDE FLANGE, 4 additional values can now be provided. The first is the width of concrete to the left of center of the steel web (b1). The second is the concrete width to the right (b2). The third is the concrete depth (d1) to be considered. The last is the modular ratio. The above values are accepted in the program by adding a ‘-’ at the first position on the first line of data. The program now awaits four
extra values on line 2 as described above. If (-) is provided on the second line the program requires another 2 breadths + 1 thickness for the bottom plate.

**D3.C.7.2 I Section**
The same is true for ISECTION definition in user table.

<table>
<thead>
<tr>
<th>D3.C.7.3 Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNIT</td>
</tr>
<tr>
<td>WIDE</td>
</tr>
<tr>
<td>C45752</td>
</tr>
<tr>
<td>-66.5</td>
</tr>
<tr>
<td>150</td>
</tr>
<tr>
<td>ISECTION</td>
</tr>
<tr>
<td>PG9144</td>
</tr>
<tr>
<td>-92.05</td>
</tr>
<tr>
<td>40</td>
</tr>
</tbody>
</table>

The larger British sections have been coded as USER TABLES under wide flange and are available on request to any existing user. Please note however that composite design is not available in this portion of STAAD.

**D3.D. British Codes - Design per BS8007**
STAAD.Pro is capable of performing concrete design based on the British code BS8007:1987 *Design of concrete structures for retaining aqueous liquids*. It is recommended that the design of the structure is carried out according to BS8110, unless modified by the recommendations given in BS8007.

The information in this section is to be used in conjunction with the BS8110. Refer to **D3.A. British Codes - Concrete Design per BS8110** (on page 1655)

**D3.D.1 Design Process**
The design process is carried out in three stages.

1. **Ultimate Limit States**

   The program is structured so that ultimate design is first carried out in accordance with recommendations given in BS8110. All active design load cases are considered in turn and a tabulated output is printed showing possible reinforcement arrangements. 12, 16, and 20 mm bars are considered with possible spacings from 100,125,150,175, and 200 mm. Within these spacings, the layout providing the closest area of steel is printed under each bar size. Longitudinal and transverse moments together with critical load cases for both hogging and sagging moments are also printed. Minimum reinforcement is in any case checked and provided in each direction. Wood & Armer moments may also be included in the design.

2. **Serviceability Limit States**

   In the second stage, flexural crack widths under serviceability load cases are calculated. The *first and every other occurring* design load case is considered as a serviceability load case and crack widths are calculated based on bar sizes and spacings proposed at the ultimate limit state check.

3. **Thermal crack widths**

   Crack widths due to longitudinal and transverse moments are calculated directly under bars, midway between and at corners. A tabulated output indicating critical serviceability load cases and moments for top and bottom of the slab is then produced.
Finally thermal, crack width calculations are carried out. Through available parameters, the user is able to provide information on the type of slab, temperature range and crack width limits.

Surface zone depths are determined based on the type of slab and critical areas of reinforcements are calculated and printed in a tabulated form.

Four bar sizes are considered and for each, max crack spacing, Smax and crack widths are calculated for the critical reinforcements and printed under each bar size.

Maximum bar spacing to limit crack widths to the user's limit is also printed under each bar size.

D3.D.2 Design Parameters

The program contains a number of parameters which are needed to perform and control the design to BS8007.

These parameters not only act as a method to input required data for code calculations but give the Engineer control over the actual design process. Default values of commonly used values for conventional design practice have been chosen as the basis. Table 2D.1 contains a complete list of available parameters with their default values.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FC</td>
<td>30 N/mm²</td>
<td>Concrete grade, in current units of length and force.</td>
</tr>
<tr>
<td>CLEAR</td>
<td>20 mm</td>
<td>Distance from the outer surface to the edge of the bar, in current units of length. This is considered the same on both surfaces.</td>
</tr>
<tr>
<td>SRA</td>
<td>0.0</td>
<td>Orthogonal reinforcement layout without considering torsional moment Mxy - slabs on orthogonal reinforcement layout with Mxy used to calculate Wood &amp; Armer moments for design. A* Skew angle considered in Wood &amp; Armer equations. A* is any angle in degrees.</td>
</tr>
<tr>
<td>SCON</td>
<td>1</td>
<td>Parameter which indicates the type of slab e.e. ground or suspended as defined in BS8007</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = Suspended Slab</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = Ground Slab</td>
</tr>
</tbody>
</table>
### Design

#### D. Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEMP</td>
<td>30°C</td>
<td>Temperature range to be considered in thermal crack width calculations</td>
</tr>
<tr>
<td>CRACK *</td>
<td>0.2 mm</td>
<td>Limiting thermal crack width, in current units of length</td>
</tr>
</tbody>
</table>

* Provided in current unit systems

#### D3.D.3 Structural Model

Structural slabs that are to be designed to BS8007 must be modeled using finite elements. Refer to [G.5 Finite Element Information](#) (on page 2308) for information on the sign convention used in the program for defining elements.

It is recommended to connect elements in such a way that the positive local z axis points outwards away from the center of the container. In this manner the "Top" of elements will consistently fall on the outer surface and internal pressure loads will act in the positive direction of the local z axis.

An example of a rectangular tank is provided to demonstrate the above procedure.

Element properties are based on the thickness given under `ELEMENT PROPERTIES` command. The following example demonstrates the required input for a 300 mm slab modeled with ten elements.

```
UNIT MM
ELEMENT PROPERTIES
1 TO 10 THI 300.0
```

#### D3.D.4 Wood & Armer Moments

This is controlled by the SRA parameter. If the default value of zero is used, the design will be based on the Mx and My moments which are the direct results of STAAD analysis. The SRA parameter (Set Reinforcement Angle) can be manipulated to introduce Wood & Armer moments into the design replacing the pure Mx, My moments. These new design moments allow the Mxy moment to be considered when designing the section. Orthogonal or skew reinforcement may be considered. SRA set to -500 will assume an orthogonal layout. If however a skew is to be considered, an angle is given in degrees, measured between the local element x axis anti-clockwise (positive). The resulting Mx* and My* moments are calculated and shown in the design format.

#### D3.E. British Codes - Design per British Cold Formed Steel Code

STAAD.Pro is capable of performing steel design based on the British code BS 5950-5:1998 *Structural use of steelwork in building - Part 5: Code of practice for design of cold formed thin gauge sections*. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations. Cold work of forming strengthening effects have been included as an option.

#### D3.E.1 Cross-Sectional Properties

You specify the geometry of the cross-section by selecting one of the section shape designations from the Gross Section Property Tables published in the “The Steel Construction Institute”, *(Design of Structures using Cold Formed Steel Sections).*
The Tables are currently available for the following shapes:

- Channel with Lips
- Channel without Lips
- Z with Lips
- Pipe
- Tube

Shape assignment may be done using the Properties - Whole Structure dialog (on page 2968) of the graphical user interface (GUI) or by specifying the section designation symbol in the input file.

The properties listed in the tables are gross section properties. STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

**D3.E.2 Design Procedure**

The following two design modes are available:

**D3.E.2.1 Code Checking**

The program compares the resistance of members with the applied load effects, in accordance with BS 5950-5:1998. Code checking is carried out for locations specified by the user via the SECTION command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. The user may choose the degree of detail in the output data by setting the TRACK parameter.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

**D3.E.2.2 Member Selection**

The user may request that the program search the cold formed steel shapes database (BS standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, presents design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

The program calculates effective section properties in accordance with Section 4 of the subject code. Cross-sectional properties and overall slenderness of members are checked for compliance with:

- Clause 6.2.2, Maximum Effective Slenderness Ratio for members in Compression
- Clause 4.2, Maximum Flat Width Ratios for Elements in Compression

**D3.E.3 Design Equations**

**D3.E.3.1 Tensile Strength**

The allowable tensile strength, as calculated in STAAD as per BS5950-5, section 7 is described below.
The tensile strength, $P_t$, of the member should be determined from clause 7.2.1

$$P_t = A_e p_y$$

where

$A_e = \text{the net area An determined in accordance with cl.3.5.4}$

$p_y = \text{the design strength}$

**D3.E.3.2 Combined bending and tension**

As per clause 7.3 of BS 5950-5:1998 members subjected to both axial tension and bending should be proportioned such that the following relationships are satisfied at the ultimate limit state

$$\frac{F_t}{P_t} + \frac{M_z}{M_{cz}} + \frac{M_y}{M_{cy}} \leq 1$$

$$M_z/M_{cz} \leq 1$$

and

$$M_y/M_{cy} \leq 1$$

where

$F_t = \text{the applies tensile strength}$

$P_t = \text{the tensile capacity determined in accordance with clause 7.2.1 of the subject code}$

$M_z, M_y, M_{cz}, M_{cy} = \text{as defined in clause 6.4.2 of the subject code}$

**D3.E.3.3 Compressive Strength**

The allowable Compressive strength, as calculated in STAAD as per BS5950-5, section 6 is described below

For sections symmetrical about both principal axes or closed cross-sections which are not subjected to torsional flexural buckling, the buckling resistance under axial load, $P_c$, may be obtained from the following equation as per clause 6.2.3 of the subject code

$$P_c = \frac{P_E P_{cs}}{\phi + \sqrt{\phi^2 - P_E P_{cs}}}$$

For sections symmetrical about a single axis and which are not subject to torsional flexural buckling, the buckling resistance under axial load, $P_c$, may be obtained from the following equation as per clause 6.2.4 of the subject code

$$P_c' = \frac{M_c P_c}{(M_c + P_c' e)}$$

Where the meanings of the symbols used are indicated in the subject clauses.

**D3.E.3.4 Torsional flexural buckling**

Design of the members which have at least one axis of symmetry, and which are subject to torsional flexural buckling should be done according to the stipulations of the clause 6.3.2 using factored slenderness ratio $\alpha L_E/r$ in place of actual slenderness ratio while reading Table 10 for the value of Compressive strength($P_c$).

where

$$\alpha = \frac{(P_E/P_{TF})}{\text{when } P_E > P_{TF}}$$

$$\alpha = I, \text{otherwise}$$

Where the meanings of the symbols used are indicated in the subject clause.
**D3.E.3.5 Combined bending and compression**

Members subjected to both axial compression and bending should be checked for local capacity and overall buckling.

Local capacity check as per clause 6.4.2 of the subject code

\[ \frac{F_c}{P_{cs}} + \frac{M_z}{M_{cz}} + \frac{M_y}{M_{cy}} \leq 1 \]

**D3.E.3.6 Overall buckling check as per clause 6.4.3 of the subject code**

For beams not subjected to lateral buckling, the following relationship should be satisfied:

\[ \frac{F_c}{P_{c}} + \frac{M_z}{C_{bx} M_{cz}} \left( 1 - \frac{F_c}{P_{Ez}} \right) + \frac{M_y}{C_{by} M_{cy}} \left( 1 - \frac{F_c}{P_{Ey}} \right) \leq 1 \]

For beams subjected to lateral buckling, the following relationship should be satisfied:

\[ \frac{F_c}{P_{c}} + \frac{M_z}{M_b} + \frac{M_y}{C_{by} M_{cy}} \left( 1 - \frac{F_c}{P_{Ey}} \right) \leq 1 \]

where

- \( F_c \) = the applied axial load
- \( P_{cs} \) = the short strut capacity as per clause 6.2.3
- \( M_z \) = the applied bending moment about z axis
- \( M_y \) = the applied bending moment about y axis
- \( M_{cz} \) = the moment capacity in bending about the local Z axis in the absence of \( F_c \) and \( M_y \), as per clause 5.2.2 and 5.6
- \( M_{cy} \) = the moment capacity in bending about the local Y axis, in the absence of \( F_c \) and \( M_z \), as per clause 5.2.2 and 5.6
- \( M_b \) = the lateral buckling resistance moment as per clause 5.6.2
- \( P_{Ez} \) = the flexural buckling load in compression for bending about the local Z axis
- \( P_{Ey} \) = the flexural buckling load in compression for bending about the local Y axis
- \( C_{bx}, C_{by} \) = taken as unity unless their values are specified by the user

\( M_{cz}, M_{cy}, M_b \) are calculated from clause numbers 5.2.2 and 5.6 in the manner described herein below.

For restrained beams, the applied moment based on factored loads should not be greater than the bending moment resistance of the section, \( M_c \)

\[ M_{cz} = S_{zz} \times p_o \]
\[ M_{cy} = S_{yy} \times p_o \]

\[ p_o = \left( 1.13 - 0.0019 \frac{D_w}{t} \right) \sqrt{\frac{Y_s}{280}} p_y \]

Where

\( p_o \) = the limiting stress for bending elements under stress gradient and should not greater than design strength \( p_y \)
For unrestrained beams the applied moment based on factored loads should not be greater than the smaller of the bending moment resistance of the section, \( M_c \), and the buckling resistance moment of the beam, \( M_b \).

Then buckling resistance moment, \( M_b \), may be calculated as follows:

\[
M_b = \frac{M_E M_y}{\phi_B + \sqrt{\phi_B^2 - M_E M_y}} \leq M_c
\]

where

\[
\phi_B = \frac{M_y + (1 + \eta)M_E}{2}
\]

\( M_Y \) = the yield moment of the section, product of design strength \( p_y \) and elastic modules of the gross section with respect to the compression flange \( Z_c \)

\( M_E \) = the elastic lateral buckling resistance as per clause 5.6.2.2

\( \eta \) = the Perry coefficient

Please refer clause numbers 5.2.2 and 5.6 of the subject code for a detailed discussion regarding the parameters used in the above mentioned equations.

The maximum shear stress should not be greater than \( 0.7 \cdot p_y \) as per clause 5.4.2

The average shear stress should not exceed the lesser of the shear yield strength, \( p_v \) or the shear buckling strength, \( q_{cr} \) as stipulated in clause 5.4.3 of the subject code.

The parameters are calculated as follows:

\[
p_v = 0.6 \cdot p_y
\]

\[
q_{cr} = (1000 \cdot t/D)^2 \text{ N/mm}^2
\]

\[
P_y = A \cdot \min(p_v, q_{cr})
\]

where

\( P_v \) = the shear capacity in N/mm\(^2\)

\( p_y \) = the design strength in N/mm\(^2\)

\( t \) = the web thickness in mm

\( D \) = the web depth in mm

For beam webs subjected to both bending and shear stresses the member should be designed to satisfy the following relationship as per the stipulations of clause 5.5.2 of the subject code

\[
(F_v/P_v)^2 + (M/M_c)^2 \leq 1
\]

where

\( F_v \) = the shear force

\( M \) = the bending moment acting at the same section as \( F_v \)

\( M_c \) = the moment capacity determined in accordance with 5.2.2

### D3.E.4 Design Parameters

The design parameters outlined in Table 2E.1 are used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application’s specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.
**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 134: British Cold Formed Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>BS5950 COLD</td>
<td>Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>When this parameter is set to 1.0 (default), the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the BEAM value is 0.0, the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (See STAAD manual for details. For TRUSS members only start and end locations are designed. )</td>
</tr>
<tr>
<td>CMZ</td>
<td>1.0</td>
<td>Coefficient of equivalent uniform bending $C_b$. See BS: 5950-5:1998,5.6. Used for Combined axial load and bending design.</td>
</tr>
<tr>
<td>CMY</td>
<td>1.0</td>
<td>Coefficient of equivalent uniform bending $C_b$. See BS: 5950-5:1998,5.6. Used for Combined axial load and bending design.</td>
</tr>
</tbody>
</table>
| CWY            | 1.0           | Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See BS: 5950-5:1998,3.4  

  0 – effect should not be included  

  1 – effect should be included |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLX</td>
<td>1</td>
<td>Specifies whether torsional-flexural buckling restraint is provided or is not necessary for the member. See BS:5950-5:1998, 5.6 Values: 0 – Section subject to torsional flexural buckling 1 – Section not subject to torsional flexural buckling</td>
</tr>
<tr>
<td>FU</td>
<td>430 MPa</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td>FYLD</td>
<td>250 MPa</td>
<td>Yield strength of steel in current units.</td>
</tr>
<tr>
<td>KX</td>
<td>1.0</td>
<td>Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor for overall buckling about the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor for overall buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LX</td>
<td>Member length</td>
<td>Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Effective length for overall buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Effective length for overall buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
</tbody>
</table>
| MAIN           | 0             | Specify the design for slenderness against the maximum slenderness as per Clause 6.2.2:  
|                |               | 0 – Do not check slenderness ratio  
|                |               | 1 – Check members resisting normal loads (180)  
|                |               | 2 - Check members resisting self-weight and wind loads (250)  
<p>|                |               | 3 - Check members resisting reversal of stress (350) |
| NSF            | 1.0           | Net section factor for tension members |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DMAX</td>
<td>2540.0 cm.</td>
<td>Maximum allowable depth. It is input in the current units of length.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual to allowable stresses</td>
</tr>
</tbody>
</table>
| TRACK          | 0             | This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are:  
0 - Prints only the member number, section name, ratio, and PASS/FAIL status.  
1 - Prints the design summary in addition to that printed by TRACK 1  
2 - Prints member and material properties in addition to that printed by TRACK 2. |

D3.E.5 Verification Example

Shown below is a verification example for reference purposes.

In this problem, we have assigned Channel sections with lips to different members. Member numbers 28 to 31 have been assigned section 230CLHS66X16, member numbers 3 to 6 and 15 to 19 have been assigned the section 230CLMIL70X30 and member numbers 1, 2, 7 to 14 have been assigned the section 170CLHS56X18. These members have been designed as per BS 5950 Part 5. Other sections have been assigned from the AISI shapes database (American cold-formed steel) and designed in accordance with that code.

Solution

A. Bending Check

As per Clause 5.2.2.2 of BS 5950 – Part 5 the limiting compressive stress, \( p_o \), for stiffened webs is given by the minimum of

\[
p_o = \left( 1.13 - 0.0019 \frac{D_w}{t} \sqrt{\frac{Y_s}{280}} \right) p_y
\]

\( p_0 = p_y \), where \( p_y = \text{Min} \left( F_{YLD} \cdot 0.84 \cdot F_U \right) = 361.2 \text{ N/mm}^2 \)

So that

\[
p_0 = [1.13 \cdot 0.0019 \cdot (170/1.8) \cdot (279.212/280)^{1/2}] \cdot 361.2 = 332.727 \text{ N/mm}^2
\]

The limiting compressive moments in local Y and Z axes will be given by

\[
M_{cz} = S_{zz} \cdot p_0 = 27,632.4(332.727) = 9.19(10)^6 \text{ N mm}
\]
\[ M_{cy} = S_{yy} \cdot p_o = 27,632.4(5,427.50) = 3.46(10)^6 \text{ N} \cdot \text{mm} \]

Maximum bending moment about local Z = 2159 N·m at node 7

Maximum bending moment about local Y = 19.755 N·m at node 7

Bending Ratio Z = \( 2.15 \times 10^6 / 9.19 \times 10^6 = 0.235 \)

Bending Ratio Y = \( 19755.3 / 3.46 \times 10^6 = 0.0057 \)

Biaxial Bending ratio = 0.235 + 0.0057 = 0.2407

Buckling resistance moment \( M_b \)

As per section 5.6.2, the buckling resistance moment

\[ M_b = \frac{M_E M_y}{\phi_B + \sqrt{\phi_B^2 - M_E M_y}} \leq M_c \]

Where:

The Yield moment of section is given by

\[ M_Y = S_{zz} \cdot p_o = 9.19(10)^6 \text{ N} \cdot \text{mm} \]

The elastic buckling resistance moment as per clause 5.6.2.2 is calculated to be

\[ M_E = 4.649(10)^6 \text{ N} \cdot \text{mm} \]

And

\[ \phi_B = \frac{M_y + (1 + \eta)M_E}{2} \]

So that

\[ \phi_B = \frac{9.19(10)^6 + (1 + 0.0)4.649(10)^6}{2} = 2.325(10)^{10} \]

Which yields

\[ M_b = \frac{4.649(10)^6 \cdot 9.19(10)^6}{2.325(10)^{10} + \sqrt{[2.325(10)^{10}]^2 - 4.649(10)^6 \cdot 9.19(10)^6}} = 9.98(10)^6 \text{ N} \cdot \text{mm} \]

B. Compression Check

The Axial force induced in member# 1 is 3,436.75 N

The elastic flexural buckling load \( P_E = 1.185(10)^6 \) N

The short strut capacity (\( P_{cs} \)) is given by

\[ A_{eff} p_y = 457.698(344) = 157,448 \text{ N} \]

Perry Coefficient (\( \eta \)) = 0.02074

\[ \phi = \frac{P_{cs} + (1 + \eta)P_E}{2} = 683,512.45 \text{ N} \]

Buckling resistance

\[ P_c = \frac{P_E P_{cs}}{\phi + \sqrt{\phi^2 - P_E P_{cs}}} = 153,782 \text{ N} \]

For Channel section (being singly symmetric), Buckling Resistance as per clause 6.2.4 is
\[ P' = \frac{M_c P_c}{(M_c + P_c e_s)} \]

Where:

The limiting compressive moment, \( M_c \), in the relevant direction is equal to 9.19(10)^6 N·mm, as calculated above.

And the distance, \( e_s \), of the geometric neutral axis of the gross cross section and that of the effective cross section is equal to 38.24 m.

So that,

\[ P' = \frac{9.19(10)^6 \cdot 153,782}{9.19(10)^6 + 153,782(38.24)} = 93,788.7 N \]

Compression ratio = 3,436.75/93,788.7 = 0.0366

C. Axial Compression and Bending

\[ \frac{F_c}{P_c} + \frac{M_z}{M_b} + \frac{M_y}{C_{by} M_{cy} \left( 1 - \frac{F_c}{P_{Ey}} \right)} \leq 1 \]

\[ 3,436.75/93,788.7 + 2.15(10)^6/(9.98(10)^6) + 19,755.3/1.81(10)^6 = 0.2647 \]

Local capacity check as per clause 6.4.2

\[ \frac{F_c}{P_{cs}} + \frac{M_z}{M_{cz}} + \frac{M_y}{M_{cy}} = \frac{3,436.75}{457.698(379.212)} + \frac{2.15(10)^6}{9.19(10)^6} + \frac{19,755.3}{1.81(10)^6} = 0.2773 \]

Overall buckling check per 6.4.3

\[ \frac{F_c}{P_c} + \frac{M_z}{C_{bx} M_{cz} \left( 1 - \frac{F_c}{P_{Ez}} \right)} + \frac{M_y}{C_{by} M_{cy} \left( 1 - \frac{F_c}{P_{Ey}} \right)} \leq 1 \]

= 0.2773

D. Shear Check as per clause 5.4.2 and 5.4.3

\[ p_v = 0.6 \quad p_y = 0.6(379.212) = 227.52 \text{ N/mm}^2 \]

\[ q_{cr} = (1000 \cdot t/D)^2 = (1000 \cdot 1.8/170)^2 = 112.11 \text{ N/mm}^2 \]

\[ P_v = A \min(p_v, q_{cr}) \]

Shear resistance Y = 33,579.4 N

Shear resistance Z = 21,148.6 N

Shear Ratio Y = 5,627.72/33,579.4 = 0.1675

Shear Ratio Z = 5,627.72/21,148.6 = 0.0031

E. Shear Check with Bending as per clause 5.5.2

Shear with bending on Z

\[ (F_v/P_v)^2 + (M_z/M_{cz})^2 = (5,627.72/33,579.4)^2 + [2.15 \cdot 10^6/(9.19 \cdot 10^6)]^2 = \]

Shear with bending on Y

\[ (F_v/P_v)^2 + (M_y/M_{cy})^2 = (67.114/21,148.6)^2 + [19,755.3/(3.46 \cdot 10^6)]^2 = \]
### Table 135: Comparison for verification problem

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Hand Calculation</th>
<th>STAAD.Pro Result</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial compression ratio</td>
<td>0.0366</td>
<td>0.037</td>
<td>none</td>
</tr>
<tr>
<td>Axial compression and bending interaction ratio (overall buckling)</td>
<td>0.2773</td>
<td>0.278</td>
<td>none</td>
</tr>
<tr>
<td>Bending Z ratio</td>
<td>0.235</td>
<td>0.236</td>
<td>none</td>
</tr>
<tr>
<td>Bending Y ratio</td>
<td>0.0057</td>
<td>0.006</td>
<td>none</td>
</tr>
<tr>
<td>Biaxial bending ratio</td>
<td>0.241</td>
<td>0.241</td>
<td>none</td>
</tr>
<tr>
<td>Shear Z ratio</td>
<td>0.1675</td>
<td>0.168</td>
<td>none</td>
</tr>
<tr>
<td>Shear Y ratio</td>
<td>0.0031</td>
<td>0.003</td>
<td>none</td>
</tr>
<tr>
<td>Bending Z and Shear Y interaction ratio</td>
<td>0.08327</td>
<td>0.084</td>
<td>none</td>
</tr>
<tr>
<td>Bending Y and Shear Z interaction ratio</td>
<td>0.000043</td>
<td>0.000</td>
<td>none</td>
</tr>
</tbody>
</table>

---

### Input File

```plaintext
STAAD SPACE
SET ECHO OFF
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
  1 0 5 0; 2 0 5 10; 3 10 5 0; 4 10 5 10; 5 5 5 0;
  6 5 5 10; 7 0 5 2; 8 0 5 4;
  9 0 5 6; 10 0 5 8; 11 10 5 2; 12 10 5 4; 13 10 5 6; 14 10 5 8; 15 5 5 2;
  16 5 5 4; 17 5 5 6; 18 5 5 8; 19 10 0 0; 20 10 0 10; 21 0 0 10; 22 0 0 0;
MEMBER INCIDENCES
  1 1 7; 2 3 11; 3 1 5; 4 2 6; 5 5 3; 6 6 4; 7 7 8;
  8 8 9; 9 9 10; 10 10 2;
  11 11 12; 12 12 13; 13 13 14; 14 14 4; 15 5 15; 16 15 16; 17 16 17; 18 17 18;
  19 18 6; 20 7 15; 21 15 11; 22 8 16; 23 16 12; 24 9 17; 25 17 13; 26 18 10;
  27 18 14; 28 1 22; 29 2 21; 30 3 19; 31 4 20; 32 1 21; 33 21 4; 34 4 19;
  35 19 1; 36 2 20; 37 20 3; 38 3 22; 39 22 2;
MEMBER PROPERTY COLDFORMED AMERICAN
  32 TO 39 TABLE ST 3LU3X060
  20 TO 27 TABLE ST 3HU3X075
MEMBER PROPERTY COLDFORMED BRITISH
  28 TO 31 TABLE ST 230CLHS66X16
  3 TO 6 15 TO 19 TABLE ST 230CLMIL70X30
  1 2 7 TO 14 TABLE ST 170CLHS56X18
UNIT MMS
PRINT MEMBER PROPERTIES LIST 32 20 28 3 1
```
SUPPORTS
19 TO 22 PINNED
UNIT FEET
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-006
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
BETA 90 MEMB 20 TO 27
MATERIAL STEEL MEMB 1 TO 39
MEMBER TENSION
32 TO 39
UNIT FEET KIP
LOAD 1 VERTICAL AND HORIZONTAL
MEMBER LOAD
3 TO 6 20 TO 27 UNI GY -0.3 0 5
JOINT LOAD
1 2 FX 0.6
2 4 FZ -0.6
PERFORM ANALYSIS PRINT STATICS CHECK
UNIT KGS CM
PRINT JOINT DISP LIST 1 4 16
PRINT SUPPORT REACTIONS
PRINT MEMBER FORCES LIST 3 24 28
UNIT KIP INCH
PARAMETER 1
CODE AISI
FYLD 55 ALL
GWY 1 ALL
BEAM 1 ALL
TRACK 2 ALL
CHECK CODE MEMB 20 21
PARAMETER 2
CODE BS5950 COLD
TRACK 2 MEMB 1 TO 19 28 TO 31
CHECK CODE MEMB 1 2
FINISH

Output
The excerpts from the design output for member number 1 are as follows:

STAAD.Pro CODE CHECKING - (BS5950-5-v1.1)
****************************
UNITS : MM, KN, KNM, MPA

- | MEMBER# 1 SECTION: 170CHSS56X18 LEN: 609.60 LOCATION: 609.60 |
  | STATUS: PASS RATIO = 0.278 GOV.MODE: 6.4-Bend + Compress GOV.LOAD: 1 |

---
MATERIAL DATA:

Yield strength of steel : 379.21 N/mm²
Ultimate tensile strength : 430.00 N/mm²

SECTION PROPERTIES:(units - cm)
Section Name : 170CLHS56X18
Member Length : 60.96
Gross Area(Ag) : 5.45 Net Area (Ae): 4.58

<table>
<thead>
<tr>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment of inertia (I) : 237.27</td>
<td>21.93</td>
</tr>
<tr>
<td>Moment of inertia (Ie) : 235.46</td>
<td>19.42</td>
</tr>
<tr>
<td>Elastic modulus (Zet) : 27.85</td>
<td>5.20</td>
</tr>
<tr>
<td>Elastic modulus (Zec) : 27.55</td>
<td>10.42</td>
</tr>
</tbody>
</table>

DESIGN DATA:

<table>
<thead>
<tr>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression Capacity (Pc) : 93.70</td>
<td></td>
</tr>
<tr>
<td>Moment Capacity (Mc) : 9.17</td>
<td>3.47</td>
</tr>
<tr>
<td>Shear Capacity (Pv) : 21.00</td>
<td>33.50</td>
</tr>
<tr>
<td>LTB Capacity (Mb) : 9.17</td>
<td></td>
</tr>
</tbody>
</table>

EACH CLAUSE CHECK UNDER CRITICAL LOAD :

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>COMBINATION</th>
<th>RATIO</th>
</tr>
</thead>
<tbody>
<tr>
<td>BS-6.3</td>
<td>Compression ratio - Axial</td>
<td>0.037</td>
</tr>
<tr>
<td>BS-6.4</td>
<td>Bend-Compression ratio</td>
<td>0.278</td>
</tr>
<tr>
<td>BS-5.1</td>
<td>Bending Ratio - Z</td>
<td>0.236</td>
</tr>
<tr>
<td>BS-5.1</td>
<td>Bending Ratio - Y</td>
<td>0.006</td>
</tr>
<tr>
<td>BS-5.1</td>
<td>Biaxial Bending Ratio</td>
<td>0.241</td>
</tr>
<tr>
<td>BS-5.4</td>
<td>Shear Ratio - Z</td>
<td>0.168</td>
</tr>
<tr>
<td>BS-5.4</td>
<td>Shear Ratio - Y</td>
<td>0.003</td>
</tr>
<tr>
<td>BS-5.5.2</td>
<td>Bending -Z &amp; Shear - Y Ratio</td>
<td>0.084</td>
</tr>
<tr>
<td>BS-5.5.2</td>
<td>Bending -Y &amp; Shear - Z Ratio</td>
<td>0.000</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.
D4.A. Canadian Codes - Concrete Design per CSA Standard A23.3-94

STAAD.Pro is capable of performing concrete design based on the Canadian code CSA A23.3 1994 *Design of Concrete Structures*. Given the width and depth (or diameter for circular columns) of a section, the program will calculate the required reinforcement to resist the forces and moments.

D4.A.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams - Prismatic (Rectangular, Square & Tee)
- For Columns - Prismatic (Rectangular, Square and Circular)
- For Slabs - 4-noded Plate Elements

D4.A.2 Member Dimensions

Concrete members that are to be designed by STAAD must have certain section properties input under the MEMBER PROPERTIES command. The following example demonstrates the required input:

```
UNIT   MM
MEMBER PROPERTIES
1 3  TO 7 9 PRISM YD 450. ZD 300.
11 14  PR YD 300.
```

In the above input, the first set of members are rectangular (450mm depth and 300mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with a 300mm diameter.

D4.A.3 Slenderness Effects and Analysis Considerations

STAAD.Pro provides the user with two methods of accounting for the slenderness effect in the analysis and design of concrete members. The first method is equivalent to the procedure presented in CSA Standard A23.3-94 Clause 10.13. STAAD.Pro accounts for the secondary moments, due to axial loads and deflections, when the PDELTA ANALYSIS command is used. After solving for the joint displacements of the structure, the program calculates the additional moments induced in the structure due to the P-Delta effect. Therefore, by performing a P-Delta analysis, member forces are calculated which will require no user modification before beginning member design. Refer to TR.37.2 P-Delta Analysis Options (on page 2797) for additional details on this analysis facility.

The second method by which STAAD allows the user to account for the slenderness effect is through user supplied moment magnification factors (see the parameter MMAG in D4.A.4 Design Parameters (on page 1697)). Here the user approximates the additional moment by supplying a factor by which moments will be multiplied before beginning member design. This second procedure allows slenderness to be considered in accordance with Clause 10.14 of the code.

**Note:** STAAD.Pro does not factor loads automatically for concrete design. All the proper factored loads must be provided by the user before the ANALYSIS specification.

While performing a P-Delta analysis, all load cases must be defined as primary load cases. If the effects of separate load cases are to be combined, it should be done either by using the REPEAT LOAD command or by specifying the load information of these individual loading cases under one single load case. Usage of the LOAD COMBINATION command will yield incorrect results for P-Delta Analysis in STAAD.Pro.
D4.A.4 Design Parameters

The program contains a number of parameters which are needed to perform design per CSA STANDARD A23.3-94. These parameters not only act as a method to input required data for code calculations but give the engineer control over the actual design process. Default values, which are commonly used numbers in conventional design practice, have been used for simplicity. Table 3A.1 contains a list of available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 136: Canadian Concrete Design CSA-A23.3-94 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CLB</td>
<td>40mm</td>
<td>Clear cover to reinforcing bar at bottom of cross section.</td>
</tr>
<tr>
<td>CLS</td>
<td>40mm</td>
<td>Clear cover to reinforcing bar along the side of the cross section.</td>
</tr>
<tr>
<td>CLT</td>
<td>40mm</td>
<td>Clear cover to reinforcing bar at top of cross section.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>Depth of the concrete member. This value defaults to YD as provided under MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>EFACE</td>
<td>0.0 Face of Support</td>
<td>Distance of face of support from end node of beam. Used for shear and torsion calculation. <strong>Note:</strong> Both SFACE &amp; EFACE must be positive numbers.</td>
</tr>
<tr>
<td>FC</td>
<td>30 N/mm$^2$</td>
<td>Specified compressive strength of concrete.</td>
</tr>
<tr>
<td>FYMAIN</td>
<td>400N/mm$^2$</td>
<td>Yield Stress for main reinforcing steel.</td>
</tr>
<tr>
<td>FYSEC</td>
<td>400 N/mm$^2$</td>
<td>Yield Stress for secondary reinforcing steel.</td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>Number 55 bar</td>
<td>Maximum main reinforcement bar size.</td>
</tr>
<tr>
<td>MINMAIN</td>
<td>Number 10 bar</td>
<td>Minimum main reinforcement bar size.</td>
</tr>
</tbody>
</table>
## Parameter Name | Default Value | Description
---|---|---
MINSEC | Number 10 bar | Minimum secondary (stirrup) reinforcement bar size.
MMAG | 1.0 | A factor by which the column design moments will be magnified.
NSECTION | 12 | Number of equally-spaced sections to be considered in finding critical moments for beam design.
REINF | 0.0 | Tied Column. A value of 1.0 will mean spiral.
SFACE | 0.0 | Distance of face of support from start node of beam. Used for shear and torsion calculation. **Note:** Both SFACE & EFACE must be positive numbers.
TRACK | 0.0 | 0. Critical Moment will not be printed out with beam design report. 1. Moments will be printed.
WIDTH | ZD | Width of the concrete member. This value defaults to ZD as provided under MEMBER PROPERTIES.

### D4.A.5 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are scanned to create moment and shear envelopes, and locate critical sections. The total number of sections considered is thirteen (start, end, and 11 intermediate), unless that number is redefined with the NSECTION parameter.

#### D4.A.5.1 Design for Flexure

Design for flexure is performed per the rules of Chapter 10 of CSA Standard A23.3-94. Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the thirteen sections. Each of these sections are designed to resist the critical sagging and hogging moments. Currently, design of singly reinforced sections only is permitted. If the section dimensions are inadequate as a singly reinforced section, such a message will be printed in the output. Flexural design of beams is performed in two passes. In the first pass, effective depths of the sections are determined with the assumption of single layer of assumed reinforcement and reinforcement requirements are calculated. After the preliminary design, reinforcing bars are chosen from the internal database in single or multiple layers. The entire flexure design is performed again in a second pass taking into account the changed effective depths of sections calculated on the basis of reinforcement provided after the preliminary design. Final provision of flexural reinforcements are made then. Efforts have been made to meet...
the guideline for the curtailment of reinforcements as per CSA Standard A23.3-94. Although exact curtailment lengths are not mentioned explicitly in the design output (which finally will be more or less guided by the detailer taking into account other practical considerations), the user has the choice of printing reinforcements provided by STAAD at 13 equally spaced sections from which the final detailed drawing can be prepared.

The following annotations apply to the output for Beam Design.

- **LEVEL**: Serial number of bar level which may contain one or more bar group.
- **HEIGHT**: Height of bar level from the bottom of beam.
- **BAR INFORMATION**: Reinforcement bar information specifying number of bars and size.
- **FROM**: Distance from the start of the beam to the start of the rebar.
- **TO**: Distance from the start of the beam to the end of the rebar.
- **ANCHOR(STA,END)**: States whether anchorage, either a hook or continuation, is needed at start (STA) or at the end (END) of the bar.

### D4.A.5.2 Design for Shear and Torsion

Design for shear and torsion is performed per the rules of Chapter 11 of CSA Standard A23.3-94. Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear design is performed at the start and end sections. The location along the member span for design is chosen as the effective depth + SFACE at the start, and effective depth + EFACE at the end. The load case which gives rise to the highest stirrup area for shear & torsion is chosen as the critical one. The calculations are performed assuming 2-legged stirrups will be provided. The additional longitudinal steel area required for torsion is reported.

The stirrups are assumed to be U-shaped for beams with no torsion, and closed hoops for beams subjected to torsion.

### D4.A.5.3 Example of Input

Example of Input Data for Beam Design

```
UNIT      NEWTON MMS
START     CONCRETE DESIGN
CODE      CANADA
FYMAIN    415 ALL
FYSEC     415 ALL
FC        35 ALL
CLEAR     25 MEMB 2 TO 6
MAXMAIN   40 MEMB 2 TO 6
TRACK     1.0 MEMB 2 TO 9
DESIGN    BEAM 2 TO 9
END       CONCRETE DESIGN
```

### D4.A.6 Column Design

Column design is performed per the rules of Chapters 7 & 8 of the CSA Standard A23.3-94. Columns are designed for axial force and biaxial moments at the ends. All active loadings are tested to calculate reinforcement. The loading which produces maximum reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and square sections, the reinforcement is always assumed to be equally distributed on each side. That means the total number of bars will always be a multiple of four (4). This may cause slightly conservative results in some cases.
Example of Input Data for Column Design

UNIT  NEWTON MMS
START  CONCRETE DESIGN
CODE  CANADIAN
FYMAIN  415 ALL
FC 35  ALL
CLEAR  25 MEMB 2 TO 6
MAXMAIN  40 MEMB 2 TO 6
DESIGN  COLUMN 2 TO 6
END  CONCRETE DESIGN

D4.A.7 Slab and Wall Design

To design a slab or wall, it must be modeled using finite elements. The commands for specifying elements are in accordance with the relevant sections of the Technical Reference Manual.

Elements are designed for the moments Mx and My using the same principles as those for beams in flexure. The width of the beam is assumed to be unity for this purpose. These moments are obtained from the element force output. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement. The effective depth is calculated assuming #10 bars are provided. The parameters FYMAIN, FC, CLT, and CLB listed in D4.A.4 Design Parameters (on page 1697) are relevant to slab design. Other parameters mentioned in Table 3A.1 are not applicable to slab design. The output consists only of area of steel required. Actual bar arrangement is not calculated because an element most likely represents just a fraction of the total slab area.

Figure 175: Element moments: Longitudinal (L) and Transverse (T)

Example of Input Data for Slab/Wall Design

UNIT  NEWTON MMS
START  CONCRETE DESIGN
CODE  CANADA
FYMAIN  415 ALL
FC 35  ALL
CLB  40 ALL
D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01

STAAD.Pro is capable of performing steel design based on the Canadian code CAN/CSA-S16-01 Limit States Design of Steel Structures.

D4.B.1 General Comments

The design of structural steel members in accordance with the specification CAN/CSA S16-01 Limit States Design of Steel Structures is can be used in STAAD.Pro. This code supercedes the previous edition of the code CAN/CSA – S16.1-94.

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-states are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the probability of limits being surpassed is acceptably low.

In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD.Pro implementation of CAN/CSA-S16-01. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D4.B.2 Analysis Methodology

The elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the specified primary and combination loading condition. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D4.B.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to the STAAD.Pro Technical Reference Manual.

D4.B.4 Built-in Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used
for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Almost all Canadian steel sections are available for input. A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Following is the description of the different types of sections available:

**D4.B.4.1 Welded Wide Flanges (WW shapes)**

Welded wide flange shapes listed in the CSA steel tables can be designated using the same scheme used by CSA. The following example illustrates the specification of welded wide flange shapes.

<table>
<thead>
<tr>
<th>100 TO 150 TA ST WW400X444</th>
</tr>
</thead>
<tbody>
<tr>
<td>34 35 TA ST WW900X347</td>
</tr>
</tbody>
</table>

**D4.B.4.2 Wide Flanges (W shapes)**

Designation of wide flanges in STAAD is the same as that in CSA tables. For example,

<table>
<thead>
<tr>
<th>10 TO 75 95 TO 105 TA ST W460X106</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 TO 200 TA ST W610X101</td>
</tr>
</tbody>
</table>

**D4.B.4.3 S, M, HP shapes**

In addition to welded wide flanges and regular wide flanges, other I shaped sections like S, M and HP shapes are also available. The designation scheme is identical to that listed in the CSA tables. While specifying the sections, it should be remembered that the portion after the decimal point should be omitted. Thus, M310X17.6 should be specified as M310X17 and S180X22.8 should be specified as S180X22. Examples illustrating specifications of these shapes are provided below.

<table>
<thead>
<tr>
<th>10 TO 20 BY 2 TA ST S510X98</th>
</tr>
</thead>
<tbody>
<tr>
<td>45 TO 55 TA ST M150X6</td>
</tr>
<tr>
<td>88 90 96 TA ST HP310X79</td>
</tr>
</tbody>
</table>

**D4.B.4.4 Channel Sections (C & MC shapes)**

C and MC shapes are designated as shown in the following example. As in S, M and HP sections, the portion after the decimal point must be omitted in section designations. Thus, MC250X42.4 should be designated as MC250X42.

<table>
<thead>
<tr>
<th>55 TO 90 TA ST C250X30</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 TO 45 TA ST MC200X33</td>
</tr>
</tbody>
</table>

**D4.B.4.5 Double Channels**

Back-to-back double channels, with or without spacing between them, are specified by preceding the section designation by the letter D. For example, a back-to-back double channel section C200X28 without any spacing in between should be specified as:

| 100 TO 120 TA D C200X28 |

If a spacing of 2.5 length units is used, the specification should be as follows:

| 100 TO 120 TA D C200X28 SP 2.5 |

Note that the specification SP after the section designation is used for providing the spacing. The spacing should always be provided in the current length unit.
### D4.B.4.6 Angles

To specify angles, the angle name is preceded by the letter L. Thus, a 200X200 angle with a 25mm thickness is designated as L200X200X25. The following examples illustrate angle specifications.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>75 TO 95 TA ST L100X100X8</td>
</tr>
<tr>
<td>33 34 35 TA ST L200X100X20</td>
</tr>
</tbody>
</table>

Note that the above specification is for "standard" angles. In this specification, the local z-axis (see Fig. 2.6 in the Technical Reference Manual) corresponds to the Y'-Y' axis shown in the CSA table. Another common practice of specifying angles assumes the local y-axis to correspond to the Y'-Y' axis. To specify angles in accordance with this convention, the reverse angle designation facility has been provided. A reverse angle may be specified by substituting the word ST with the word RA. Refer to the following example for details.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 TO 15 TA RA L55X35X4</td>
</tr>
</tbody>
</table>

The local axis systems for STANDARD and REVERSE angles is shown in Fig. 2.6 of the STAAD Technical Reference manual.

### D4.B.4.7 Double Angles

To specify double angles, the specification ST should be substituted with LD (for long leg back-to-back) or SD (short leg back-to-back). For equal angles, either SD or LD will serve the purpose. Spacing between angles may be provided by using the word SP followed by the value of spacing (in current length unit) after section designation.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 35 45 TA LD L150X100X16</td>
</tr>
<tr>
<td>80 TO 90 TA SD L125X75X6 SP 2.5</td>
</tr>
</tbody>
</table>

The second example above describes a double angle section consisting of 125X75X6 angles with a spacing of 2.5 length units.

### D4.B.4.8 Tees

Tee sections obtained by cutting W sections may be specified by using the T specification instead of ST before the name of the W shape. For example:

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 TO 120 TA T W200X42</td>
</tr>
</tbody>
</table>

will describe a T section cut from a W200X42 section.

### D4.B.4.9 Rectangular Hollow Sections

These sections may be specified in two possible ways. Those sections listed in the CSA tables may be specified as follows.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>55 TO 75 TA ST TUB80X60X4</td>
</tr>
</tbody>
</table>
In addition, any tube section may be specified by using the DT (for depth), WT (for width), and TH (for thickness) specifications. For example:

100 TO 200 TA ST TUBE DT 8.0 WT 6.0 TH 0.5

will describe a tube with a depth of 8 in., width of 6 in. and a wall thickness of 0.5 inches. Note that the values of depth, width and thickness must be provided in current length unit.

**D4.B.4.10 Circular Hollow Sections**

Sections listed in the CSA tables may be provided as follows:

15 TO 25 TA ST PIP33X2.5

In addition to sections listed in the CSA tables, circular hollow sections may be specified by using the OD (outside diameter) and ID (inside diameter) specifications. For example:

70 TO 90 TA ST PIPE OD 10.0 ID 9.0

will describe a pipe with an outside diameter of 10 length units and inside diameter of 9.0 length units. Note that the values of outside and inside diameters must be provided in terms of current length unit.

**D4.B.4.11 Example**

Sample input file to demonstrate usage of Canadian shapes

```plaintext
STAAD SPACE
UNIT METER KNS
JOINT COORD
1 0 0 0 17 160 0 0
MEMBER INCIDENCES
1 1 2 16
UNIT CM
```
D4.B.5 Section Classification

The CSA specification allows inelastic deformation of section elements. Thus, local buckling becomes an important criterion. Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11.2 and Table 1 of CAN/CSA-S16-01). This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro determines the section classification for the standard shapes and user specified shapes.

**Note:** The design of Class 4 sections requires STAAD.Pro V8i (SELECTseries 2) build 2007.07 or higher. Otherwise, design is performed for sections that fall into the category of Class 1, 2 or 3 sections only.

D4.B.6 Member Resistances

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes
into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

**Note:** The design of Class 4 sections requires STAAD.Pro V8i (SELECTseries 2) build 2007.07 or higher.

**D4.B.6.1 Nomenclature**

where

\[ A = \text{Area.} \]
\[ A_e = \text{Effective area.} \]
\[ A_f = \text{Area of flange.} \]
\[ A_w = \text{Area of web.} \]
\[ b_e = \text{Effective Flange width.} \]
\[ C_f = \text{Compressive force in a member or component under factored load.} \]
\[ C_r = \text{Factored compressive resistance.} \]
\[ C_w = \text{Warping torsional constant.} \]
\[ C_y = \text{Axial compressive load at yield stress.} \]
\[ D = \text{Outside diameter of pipe section.} \]
\[ E = \text{Elastic modulus of steel.} \]
\[ F_e = \text{Elastic critical buckling stress.} \]
\[ F_y = \text{Yield strength.} \]
\[ F_{ye} = \text{Effective yield stress of section in compression to account for elastic local buckling.} \]
\[ h = \text{Clear depth of web.} \]
\[ K = \text{Effective length factor.} \]
\[ L = \text{Length or span of member.} \]
\[ M_f = \text{Bending moment in a member or component under factored load.} \]
\[ M_r = \text{Factored moment resistance of a member.} \]
\[ M_y = \text{Yield moment resistance.} \]
\[ S = \text{Elastic section modulus.} \]
\[ S_e = \text{Effective section modulus.} \]
\[ W = \text{Web thickness.} \]
\[ \lambda = \text{Non-dimensional slenderness parameter in column formula.} \]
\[ \lambda_{ye} = \text{Effective non-dimensional slenderness parameter in column formula considering effective yield stress.} \]
\[ \phi = \text{Resistance factor} \]

**D4.B.6.2 Members Subject to Axial Forces**

**Axial Tension**

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (see Table 3B.1). STAAD calculates the tension capacity of a member based on these two limits states per Cl.13.2 of CAN/CSA-S16-01. Parameters FYLD, FU, and NSF are applicable for these calculations.

**Axial Compression**

The compressive resistance of columns is determined based on Clause 13.3 of the code. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (KL/r ratios).
The effective length for the calculation of compression resistance may be provided through the use of the parameters $K_T$, $K_Y$, $K_Z$, $L_T$, $L_Y$, and $L_Z$ (see Table 3B.1). Some of the aspects of the axial compression capacity calculations are:

1. For frame members not subjected to any bending, and for truss members, the axial compression capacity in general column flexural buckling is calculated from CL13.3.1 using the slenderness ratios for the local Y-Y and Z-Z axis. The parameters $K_Y$, $L_Y$, $K_Z$, and $L_Z$ are applicable for this.

2. For single angles, which are frame members not subjected to any bending or truss members, the axial compression capacity in general column flexural buckling and local buckling of thin legs is calculated using the rules of the AISC - LRFD code, 2nd ed., 1994. The reason for this is that the Canadian code doesn’t provide any clear guidelines for calculating this value. The parameters $K_Y$, $L_Y$, $K_Z$, and $L_Z$ are applicable for this.

3. The axial compression capacity is also calculated by taking flexural-torsional buckling into account. The rules of Appendix D, page 1-109 of CAN/CSA-S16-01 are used for this purpose. Parameters $K_T$ and $L_T$ may be used to provide the effective length factor and effective length value for flexural-torsional buckling. Flexural-torsional buckling capacity is computed for single channels, single angles, Tees and Double angles.

4. The variable “n” in CL13.3.1 is assumed as 2.24 for WWF shapes and 1.34 for all other shapes.

5. While computing the general column flexural buckling capacity of sections with axial compression + bending, the special provisions of 13.8.1(a), 13.8.1(b) and 13.8.1(c) are applied. For example, $\lambda = 0$ for 13.8.1(a), $K=1$ for 13.8.1(b), etc.)

For Class 4 members subjected to axial compression, factored compressive resistance should be determined by either of the following equations.

a. $C_r = \phi A_e F_y (1+\lambda^{2n})^{-1/n}$

where

\[
\begin{align*}
 n &= 1.34 \\
 \lambda &= \sqrt{F_y/F_e} \\
 F_e &= \frac{\pi^2 E}{(KL/r)^2}
\end{align*}
\]

$A_e$ is calculated using reduced element widths meeting the maximum width to thickness ratio specified in Table 1.

Effective width required for the calculation of effective area $A_e$, for different section shapes are as follows.

- For flanges of I-section, T-section and channel section and legs of angle section

  $b_e = 200t/\sqrt{F_y}$

- For stem of T-section

  $b_e = 340t/\sqrt{F_y}$

- For flanges of HSS rectangular or Tube sections

  $b_e = 670t/\sqrt{F_y}$

- For circular HSS or Pipe section

  $D = 23000t/F_y$

b. $C_r = \phi F_{ye} (1+\lambda_{ye}^{2n})^{-1/n}$

where

\[
\begin{align*}
 n &= 1.34 \\
 \lambda_{ye} &= \sqrt{F_{ye}/F_e} \\
 F_e &= \frac{\pi^2 E}{(KL/r)^2}
\end{align*}
\]

With an effective yield stress, $F_{ye}$, determined from the maximum width (or diameter)-to-thickness ratio meeting the limit specified in Table 1.
Following are the expressions for effective yield stress for different shaped section.

- For I-section, T-section, channel section and angle section
  \[ F_{ye} = \frac{40000}{(b/t)^2} \]
- For rectangular HSS section
  \[ F_{ye} = \frac{448900}{(b/t)^2} \]
- For circular HSS section
  \[ F_{ye} = \frac{23000}{(D/t)} \]

**D4.B.6.3 Members Subject to Bending**

The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified in STAAD with the help of the parameter UNL. If UNL is less than one tenth the member length (member length is the distance between the joints of the member), the member is treated as being continuously laterally supported. In this case, the moment resistance is computed from Clause 13.5 of the code. If UNL is greater than or equal to one tenth the member length, its value is used as the laterally unsupported length. The equations of Clause 13.6 of the code are used to arrive at the moment of resistance of laterally unsupported members. Some of the aspects of the bending capacity calculations are:

1. The weak axis bending capacity of all sections except single angles is calculated as
   For Class 1 & 2 sections, \( \phi \cdot P_y \cdot F_{y} \)
   For Class 3 sections, \( \phi \cdot S_y \cdot F_{y} \)
   where
   - \( \phi \) = Resistance factor = 0.9
   - \( P_y \) = Plastic section modulus about the local Y axis
   - \( S_y \) = Elastic section modulus about the local Y axis
   - \( F_{y} \) = Yield stress of steel

2. For single angles, the bending capacities are calculated for the principal axes. The specifications of Section 5, page 6-283 of AISC-LRFD 1994, 2nd ed., are used for this purpose because the Canadian code doesn't provide any clear guidelines for calculating this value.

3. For calculating the bending capacity about the Z-Z axis of singly symmetric shapes such as Tees and Double angles, CAN/CSA-S16-01 stipulates in Clause 13.6(d), page 1-31, that a rational method, such as that given in SSRC’s Guide to Stability Design Criteria of Metal Structures, be used. Instead, STAAD uses the rules of Section 2c, page 6-55 of AISC-LRFD 1994, 2nd ed.

Laterally Supported Class 4 members subjected to bending

i. When both the web and compressive flange exceed the limits for Class 3 sections, the member should be considered as failed and an error message will be thrown.

ii. When flanges meet the requirements of Class 3 but web exceeds the limits for Class 3, resisting moment shall be determined by the following equation.

\[
M' = M_r \left[ 1 - 0.0005 \frac{A_w}{A_f} \left( \frac{b}{h} - \frac{1}{\sqrt{M_f/\phi_s}} \right) \right]
\]

Where \( M_r = \) factored moment resistance as determined by Clause 13.5 or 13.6 but not to exceed \( \phi M_y = \) factored moment resistance for Class 3 sections = \( \phi M_y \)

If axial compressive force is present in addition to the moment, modified moment resistance should be as follows.
\[ M' = M_r \left\{ 1 - 0.0005 \frac{A_w}{A_f} \left[ \frac{h}{w} - 1, 900 \left( 1 - 0.65 \frac{C_f}{(\phi C_y)} \right) \right] \right\} \]

\[ C_y = A \cdot F_y \]

\( S = \text{Elastic section modulus of steel section.} \)

**iii.** For sections whose webs meet the requirements of Class 3 and whose flanges exceed the limit of Class 3, the moment resistance shall be calculated as

\[ M_r = \phi \cdot S_e \cdot F_y \]

Where:

\( S_e = \text{effective section modulus determined using effective flange width.} \)

- For Rectangular HSS section, effective flange width
  \[ b_e = 670 \cdot t/\sqrt(F_y) \]
- For I-section, T-section, Channel section, effective flange width and for Angle section, effective length width
  \[ b_e = 200 \cdot t/\sqrt(F_y) \]

But shall not exceed 60 \( \cdot t \)

Laterally Unsupported Class 4 members subjected to bending

As per clause 13.6(b) the moment resistance for class-4 section shall be calculated as follows

**i.** When \( M_u > 0.67M_y \)

\[ M_r = 1.15\phi M_y \left( 1 - 0.28 \frac{M_y}{M_u} \right) \]

\( M_r \) should not exceed \( \phi S_e F_y \)

**ii.** When \( M_u \leq 0.67 M_y \)

\[ M_r = \phi M_u \]

Where, as per clause 13.6(a),

\[ M_u = (\omega_2 n/L) \sqrt{(EI_y GJ + (nE/L)^2 I_y C_w)} \]

For unbraced length subjected to end moments-

\[ \omega_2 = 1.75 + 1.05k + 0.3k^2 \leq 2.5 \]

When bending moment at any point within the unbraced length is larger than the larger end moment or when there is no effective lateral support for the compression flange at one of the ends of unsupported length-

\[ \omega_2 = 1.0 \]

\( k = \text{Ratio of the smaller factored moment to the larger moment at opposite ends of the unbraced length, positive for double curvature and negative for single curvature.} \)

\( S_e = \text{effective section modulus determined using effective flange width.} \)

- For Rectangular HSS section, effective flange width
  \[ b_e = 670t/\sqrt(F_y) \]
- For I-section, T-section, Channel section, effective flange width and for Angle section, effective length width
\[ b_e = \frac{200t}{\sqrt{F_y}} \]

But shall not exceed 60t.

This clause is applicable only for I shaped and Channel shaped section as there is no guideline in the code for other sections.

**D4.B.6.4 Members Subject to Combined Forces**

**Axial compression and bending**

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. In these equations, the additional bending caused by the action of the axial load is accounted for by using amplification factors. Clause 13.8 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceed 1.0 or the allowable value provided using the RATIO parameter (Refer to **D4.B.7 Design Parameters** on page 1710), the member is considered to have failed under the loading condition.

**Axial tension and bending**

Members subjected to axial tension and bending are also designed using interaction equations. Clause 13.9 of the code is used to perform these checks. The actual RATIO is determined as the value of the left hand side of the critical equation.

**D4.B.6.5 Shear**

The shear resistance of the cross section is determined using the equations of Clause 13.4 of the code. Once this is obtained, the ratio of the shear force acting on the cross section to the shear resistance of the section is calculated. If any of the ratios (for both local Y & Z axes) exceed 1.0 or the allowable value provided using the RATIO parameter (see Table 3B.1), the section is considered to have failed under shear. The code also requires that the slenderness ratio of the web be within a certain limit (See Cl.13.4.1.3, page 1-29 of CAN/CSA-S16-01). Checks for safety in shear are performed only if this value is within the allowable limit. Users may by-pass this limitation by specifying a value of 2.0 for the MAIN parameter.

**D4.B.7 Design Parameters**

The design parameters outlined in Table 3B.1 may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.
### Table 137: Canadian Steel Design CSA-S16-01 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as CANADIAN 2001. Design code to follow. See [TR.48.1 Parameter Specifications](on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = design only for end moments and those at locations specified by SECTION command. 1.0 = Perform design for moments at twelfth points along the beam.</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Greater than 0.0 and less than 2.5: Value of Omega_2 (Cl.13.6) to be used for calculation. Equal to 0.0: Calculate Omega_2</td>
</tr>
<tr>
<td>CMY</td>
<td>1.0</td>
<td>1.0 = Do not calculate Omega-1 for local Y axis. 2.0 = Calculate Omega-1 for local Y axis. Used in Cl.13.8.4 of code</td>
</tr>
<tr>
<td>CMZ</td>
<td>1.0</td>
<td>1.0 = Do not calculate Omega-1 for local Z axis. 2.0 = Calculate Omega-1 for local Z axis. Used in Cl.13.8.4 of code</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>“Deflection Length”/Maxm. Allowable local deflection.</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting start point for calculation of “deflection length”</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of “deflection length”</td>
</tr>
<tr>
<td>DMAX</td>
<td>45.0 in.</td>
<td>Maximum allowable depth (Applicable for member selection)</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in.</td>
<td>Minimum required depth (Applicable for member selection)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>FU</td>
<td>345.0 MPa</td>
<td>Ultimate strength of steel.</td>
</tr>
<tr>
<td>FYLD</td>
<td>300.0 MPa</td>
<td>Yield strength of steel.</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td>K value for flexural torsional buckling.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.</td>
</tr>
<tr>
<td>LT</td>
<td>Member Length</td>
<td>Length for flexural torsional buckling.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Length for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.</td>
</tr>
<tr>
<td>MAIN</td>
<td>0.0</td>
<td>0.0 = Check slenderness ratio against the limits. 1.0= Suppress the slenderness ratio check. 2.0 = Check slenderness ratio only for column buckling, not for web (See Section 3B.6, Shear)</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual load effect to the design strength.</td>
</tr>
<tr>
<td>SHEAR</td>
<td>1</td>
<td>Shear stress calculation option. 0) = compute the actual shear stress using VQ/1b 1) = compute the actual shear stress using V/(Ay or Az)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| SSY            | 0             | 0 = Sway or Unbraced along local Y axis  
1 = Braced along local Y axis  
This parameter is required to choose the proper value of $U_{1y}$ which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength. |
| SSZ            | 0             | 0 = Sway or Unbraced along local Z axis  
1 = Braced along local Z axis  
This parameter is required to choose the proper value of $U_{1z}$ which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength. |
| TRACK          | 0.0           | 0.0 = Report only minimum design results.  
1.0 = Report design strengths also.  
2.0 = Provide full details of design. |
| UNB            | Member Length | Unsupported length in bending compression of the bottom flange for calculating moment resistance. |
| UNT            | Member Length | Unsupported length in bending compression of the top flange for calculating moment resistance. |

**D4.B.8 Code Checking**

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per the CAN/CSA-S16-01 requirements.

Code checking is done using forces and moments at specified sections of the members. If the BEAM parameter for a member is set to 1, moments are calculated at every twelfth point along the beam. When no sections are specified and the BEAM parameter is set to zero (default), design will be based on member start and end forces only. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. The extent of detail of the output can be controlled by using the TRACK parameter.
Example of commands for **CODE CHECKING**:

```
UNIT NEWTON METER
PARAMETER
CODE CANADIAN
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
UNL 15 MEMB 3 4
RATIO 0.9 ALL
CHECK CODE MEMB 3 4
```

**D4.B.9 Member Selection**

The member selection process basically involves determination of the least weight member that passes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on TUBES, PIPES or members listed as PRISMATIC.

Example of commands for **MEMBER SELECTION**:

```
UNIT NEWTON METER
PARAMETER
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
UNL 15 MEMB 3 4
RATIO 0.9 ALL
SELECT MEMB 3 4
```

**D4.B.10 Tabulated Results of Steel Design**

Results of code checking and member selection are presented in a tabular format. The term **CRITICAL COND** refers to the section of the CAN/CSA-S16-01 specification which governed the design.

If the TRACK parameter is set to 1.0, factored member resistances will be printed. Following is a description of some of the items printed.

- **CR**: Factored compressive resistance
- **TR**: Factored tensile resistance
- **VR**: Factored shear resistance
- **MRZ**: Factored moment resistance (about z-axis)
- **MRY**: Factored moment resistance (about y-axis)

Further details can be obtained by setting TRACK to 2.0.

- **CR1**: CAPACITY ($C_r$) PER 13.8.2(a)
D4.C. Canadian Codes - Cold Formed Steel Design per S136-94

STAAD.Pro is capable of performing steel design based on the Canadian code S136-94 Specification for the Design of Cold-Formed Steel Structural Members, including revisions dated May, 1995. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations. For laterally supported members in bending, the Initiation of Yielding method has been used. Cold work of forming strengthening effects have been included as an option.

D4.C.1 Cross-Sectional Properties

You specify the geometry of the cross-section by selecting one of the section shape designations from the Gross Section Property Tables published in the "Cold-Formed Steel Design Manual", AISI, 1996 Edition.

The Tables are currently available for the following shapes:

- Channel with Lips
- Channel without Lips
- Angle with Lips
- Angle without Lips
- Z with Lips
- Z without Lips
- Hat

Shape selection may be done using the member property pages of the graphical user interface (GUI) or by specifying the section designation symbol in the input file.

The properties listed in the tables are gross section properties. STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

D4.C.2 Design Procedure

The following two design modes are available:

D4.C.2.1 Code Checking

The program compares the resistance of members with the applied load effects, in accordance with CSA 136. Code checking is carried out for locations specified via the SECTION command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. You may choose the degree of detail in the output data by setting the TRACK parameter.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.
**D4.C.2.2 Member Selection**

You may request that the program search the cold formed steel shapes database (AISI standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, present design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

**D4.C.2.3 Code Sections Implemented**

The program calculates effective section properties in accordance with Clauses 5.6.2.1 through 3 and 5.6.2.6 through 8. Cross-sectional properties and overall slenderness of members are checked for compliance with

- Clause 5.3, Maximum Effective Slenderness Ratio for members in Compression
- Clause 5.4, Maximum Flat Width Ratios for Elements in Compression
- Clause 5.5, Maximum Section Depths.

The program will check member strength in accordance with Clause 6 of the Standard as follows:

- Resistance factors listed in Clauses 6.2 (a), (b), and (e) are used, as applicable.
- Members in tension - Resistance is calculated in accordance with Clauses 6.3.1 and 6.3.2.
- Members in bending and shear

Resistance calculations are based on Clauses:

- 6.4.1 General,
- 6.4.2 and 6.4.2.1 Laterally Supported Members, compressive limit stress based on Initiation of Yielding,
- 6.4.3 Laterally Unsupported Members,
- 6.4.4 Channels and Z-Shaped Members with Unstiffened Flanges - additional limitations,
- 6.4.5 Shear in Webs,
- 6.4.6 Combined Bending and Shear in Webs.
- Members in compression

Resistance calculations are based on Clauses:

- 6.6.1.1, 6.6.1.2 (a) and (d), and 6.6.1.3 General,
- 6.6.2 Sections Not Subject to Torsional-Flexural Buckling,
- 6.6.3 Singly Symmetric Sections,
- 6.6.4 Point-Symmetric Sections,
- 6.6.5 Cylindrical Tubular Sections.
- Members in compression and bending

Resistance calculations are based on Clause 6.7.1, Singly and Doubly Symmetric Sections. Input for the coefficients of uniform bending must be provided.

**D4.C.3 Design Parameters**

The following table contains the input parameters for specifying values of design variables and selection of design options.
**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 138: Canadian Cold Formed Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified S136. Design code to follow. See <a href="#">TR.48.1 Parameter Specifications</a> (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>When this parameter is set to 1.0 (default), the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the BEAM value is 0.0, the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (See STAAD manual for details). If neither the BEAM parameter nor any SECTION command is specified, STAAD will terminate the run and ask the user to provide one of those 2 commands. This rule is not enforced for TRUSS members.</td>
</tr>
<tr>
<td>CMZ</td>
<td>1.0</td>
<td>Coefficient of equivalent uniform bending $w_z$. See CSA 136, 6.7.2. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.</td>
</tr>
<tr>
<td>CMY</td>
<td>0.0</td>
<td>Coefficient of equivalent uniform bending $w_y$. See CSA 136, 6.7.2. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.</td>
</tr>
<tr>
<td>CWY</td>
<td>0</td>
<td>Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See CSA 136, 5.2.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. effect should not be included</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. effect should be included</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000.0</td>
<td>Maximum depth permissible for the section during member selection. This value must be provided in the current units.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0</td>
<td>Minimum depth required for the section during member selection. This value must be provided in the current units.</td>
</tr>
<tr>
<td>FLX</td>
<td>1</td>
<td>Specifies whether torsional-flexural buckling restraint is provided or is not necessary for the member. See CSA 136, 6.6.2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Section subject to torsional flexural buckling and restraint not provided</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. restraint provided or unnecessary</td>
</tr>
<tr>
<td>FU</td>
<td>450 MPa</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td>FYLD</td>
<td>350 MPa</td>
<td>Yield strength of steel in current units.</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td>Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor for overall column buckling about the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor for overall column buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LT</td>
<td>Member length</td>
<td>Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Effective length for overall column buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Effective length for overall column buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members, See CSA 136, 6.3.1.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Member length</td>
<td>Spacing in the longitudinal direction of shear stiffeners for stiffened flat webs. It is input in the current units of length. See section CSA 136, 6.4.5</td>
</tr>
</tbody>
</table>
### Design

#### D. Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| TRACK          | 0             | This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are:  
0. Prints only the member number, section name, ratio, and PASS/FAIL status.  
1. Prints the design summary in addition to that printed by TRACK 1  
2. Prints member and material properties in addition to that printed by TRACK 2. |
| TSA            | 1             | Specifies whether bearing and intermediate transverse stiffeners satisfy the requirements of CSA 136, 6.5. If true, the program uses the more liberal set of interaction equations in 6.4.6.  
0. stiffeners do not comply with 6.5  
1. stiffeners comply with 6.5 |

#### D4.D. Canadian Codes - Timber Design per CAN/CSA-086-01

STAAD.Pro is capable of performing timber design based on the Canadian code CSA 086-01 Wood Design Standard.

#### D4.D.1 General Comments

The design philosophy of this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for the entire structure under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD.Pro implementation, the code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD implementation of CSA086-01. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.
D4.D.2 Analysis Methodology

Analysis is done for the primary and combination loading conditions provided by the user. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations.

D4.D.3 Member Property Specifications

A timber section library consisting of Sawn and Glulam timber is available for member property specification. For specification of member properties, for Sawn timber the timber section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in timber table.

For Glulam timber, member properties can be specified using the YD (depth) and ZD (width) specifications and selecting Combination and Species specifications from the built-in table. The assignment is done with the help of the PRISMATIC option (Refer to TR.20.2 Prismatic Property Specification (on page 2465))

D4.D.4 Built-in Timber Section Library

The following information is provided for use when the built-in timber tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design.

Following are the description of the different types of species combination available:

D4.D.4.1 Douglas Fir-Larch

The following example illustrates the specification of Douglas Fir-Larch species combination.

100 TO 150 TABLE ST DFL_SelStr_2X2_BM

D4.D.4.2 Hem-Fir

Designation of Hem-Fir species combination in STAAD is as follows.

100 TO 150 TABLE ST Hem-Fir_SelStr_2X10_BM

D4.D.4.3 Northern Species

Designation of Northern species combination in STAAD is as follows.

100 TO 150 TABLE ST Northern_SelStr_3X12_BM
**D4.D.4.4 Spruce-Pine-Fir**

Designation of Spruce-Pine-Fir species combination in STAAD is as follows.

```
100 TO 150 TABLE ST SPF_SelStr_3X8_BM
```

**D4.D.4.5 Glue Laminated timber**

Designation of glue-laminated (glu-lam) timber in STAAD.Pro involves defining the material, specifying the dimensions, and associating the material with the member through the CONSTANTS command.

```
UNIT CM KN
DEFINE MATERIAL START
ISOTROPIC GLT_D.Fir-L-24f-EX
E 51611.7
POISSON 0.15
DENSITY 2.5e-005
ALPHA 1.2e-011
END DEFINE MATERIAL
MEMBER PROPERTY TIMBER CANADIAN
1 PRIS YD 12 ZD 6
CONSTANTS
MATERIAL GLT_D.Fir-L-24f-EX MEMB 1
```

**D4.D.4.6 Example**

Sample input file to demonstrate usage of Canadian timber

```
STAAD PLANE EXAMPLE FOR DIMENSIONAL LUMBER
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 6 0 0; 3 12 0 0; 4 18 0 0;
5 24 0 0; 6 6 3 0; 7 12 6 0; 8 18 3 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 1 6; 6 6 7; 7 7 8; 8 8 5;
9 2 6; 10 3 7; 11 4 8; 12 6 3; 13 3 8;
UNIT FEET POUND
DEFINE MATERIAL START
ISOTROPIC SPF_SelStr_4X10_BM
E 1224
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY tim can
1 TO 4 9 TO 11 TABLE ST SPF_SelStr_4X10_BM
5 TO 8 12 13 TABLE ST SPF_SelStr_4X10_BM
CONSTANTS
MATERIAL SPF_SelStr_4X10_BM memb 1 TO 4 9 TO 11
```
D4.D.5 Member Resistance

The member resistances are calculated in STAAD according to the procedures outlined in section 5 (for sawn lumber) and 6 (for Glulam) of CSA086-01.

These depend on several adjustment factors as follows:

- **KD**: Load duration factor (Clause 4.3.2.2-CSA086-01, Table 4.3.2.2)
- **KH**: System factor (Clause 5.4.4 and 6.4.3 and Table 5.4.4-CSA086-01)
- **K_T**: Treatment factor (Clause 5.4.3 and 6.4.4-CSA086-01)
- **KSB**: Service condition factor applicable to Bending at extreme fibre (Table 5.4.2 and 6.4.2-CSA086-01)
- **KSV**: Service condition factor applicable to longitudinal shear (Table 5.4.2 and 6.4.2 CSA086-01)
- **KSC**: Service condition factor applicable to Compression parallel to the grain (Table 5.4.2 and 6.4.2 CSA086-01)
- **K_SCP**: Service condition factor applicable to Compression perpendicular to the grain (Table 5.4.2 and 6.4.2 CSA086-01)
- **K_SE**: Service condition factor applicable to modulus of elasticity (Table 5.4.2 and 6.4.2 CSA086-01)
- **KST**: Service condition factor applicable to tension parallel to the grain (Table 5.4.2 and 6.4.2 CSA086-01)
- **KZB**: Size factor applicable to bending (Clause 5.4.5 and Table 5.4.5-CSA086-01)
- **KZV**: Size factor applicable to shear (Clause 5.4.5 and Table 5.4.5-CSA086-01)
- **KZT**: Size factor applicable to tension parallel to grain (Clause 5.4.5 and Table 5.4.5-CSA086-01)
- **KZCP**: Size factor applicable to compression perpendicular to grain (Clause 5.4.5 and Table 5.4.5-CSA086-01)
- **K_ZC**: Size factor applicable to compression parallel to grain (Clause 5.4.5 and Table 5.4.5-CSA086-01)
- **CHIX**: Curvature factor (Clause 6.5.6.5.2-CSA086-01)
- **CV**: Shear load coefficient (Table 6.5.7.4A-CSA086-01)
- **KN**: Notch factor (Clause 5.5.5.4-CSA086-01)

All of these factors must be specified as input according to the classification of timber and stress grade.

Explained here is the procedure adopted in STAAD for calculating the member resistances.

### D4.D.5.1 Axial Tension

**i. For Sawn timber**

The criterion governing the capacity of tension members is based on one limit state. The limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (see Table 3B.1). STAAD calculates the tension capacity of a member based on this limit state per Clause 5.5.9 of CSA086-01.

**ii. For Glulam timber**
The design of glulam tension members differs from sawn timber since CSA 086-01 assigns different specified strength for gross and net section. The specified strength at net section is slightly higher than the strength of the gross section. Therefore, Glulam tension members are designed based on two limit states. The first one is the limit state of yielding in the gross section. The second limit state involves fracture at the section with the minimum effective net area. The net-section area may be specified by the user through the use of the parameter NSF (see Table 3B.1). STAAD calculates the tension capacity of a member based on these two limits states per Clause.6.5.11 of CSA086-01.

D4.D.5.2 Axial Compression
The compressive resistance of columns is determined based on Clause.5.5.6 and Clause.6.5.8.4 of CSA086-01. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (Kc). The effective length for the calculation of compression resistance may be provided through the use of the parameters KX, KY, KZ, LX, LY and LZ (see Table 3B.1).

D4.D.5.3 Bending
The bending resistance of Sawn members are determined based on Clause 5.5.4 of CSA086-01 and for glulam members are determined based on Clause 6.5.6.5 of CSA086-01. The allowable stress in bending is multiplied by Lateral stability factor, KL to take in account whether lateral support is provided at points of bearing to prevent lateral displacement and rotation

D4.D.5.4 Axial compression and bending
The member strength for sections subjected to axial compression and uni-axial or biaxial bending is obtained through the use of interaction equations. Clause 5.5.10 and 6.5.12 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceeds 1.0 or the allowable value provided using the RATIO parameter (see Table 3B.1), the member is considered to have Failled under the loading condition.

D4.D.5.5 Axial tension and bending
The member strength for sections subjected to axial tension and uniaxial or biaxial bending is obtained through the use of interaction equations. Clause 5.5.10 and 6.5.12 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceeds 1.0 or the allowable value provided using the RATIO parameter (see Table 3B.1), the member is considered to have Failled under the loading condition.

D4.D.5.6 Shear
The shear resistance of the cross section is determined using the equations of Clause 5.5.5 and 6.5.7.2 of the code. Once this is obtained, the ratio of the shear force acting on the cross section to the shear resistance of the section is calculated. If any of the ratios (for both local Y & Z axes) exceed 1.0 or the allowable value provided using the RATIO parameter (see Table 3B.1), the section is considered to have failed under shear.

D4.D.6 Design Parameters
The design parameters outlined in Table below may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allows the engineer to control the design process to suit an application’s specific needs.
The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 139: Canadian Timber Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as TIMBER CANADIAN</td>
</tr>
<tr>
<td>CHIX</td>
<td>1.0</td>
<td>Curvature Factor for Compression [Clause 6.5.6.5.2]</td>
</tr>
<tr>
<td>CV</td>
<td>1.0</td>
<td>Shear Load Coefficient [Table 6.5.7.4A]</td>
</tr>
<tr>
<td>KD</td>
<td>1.0</td>
<td>Load Duration Factor [Clause 4.3.2, Table 4.3.2]</td>
</tr>
<tr>
<td>KH</td>
<td>1.0</td>
<td>System Factor [Clause 5.4.4/6.4.3, Table 5.4.4]</td>
</tr>
<tr>
<td>KN</td>
<td>1.0</td>
<td>Notch Factor [Clause 5.4.7.2.2]</td>
</tr>
<tr>
<td>KSB</td>
<td>1.0</td>
<td>Service Condition Factor for Bending at Extreme Fibre, Applicable for bending at extreme fibre [Table 5.4.2 and 6.4.2]</td>
</tr>
<tr>
<td>KSC</td>
<td>1.0</td>
<td>Service Condition Factor for Compression, Applicable for compression parallel to grain [Table 5.4.2 and 6.4.2]</td>
</tr>
<tr>
<td>KSE</td>
<td>1.0</td>
<td>Service Condition Factor for Modulus of Elasticity, Applicable for modulus of elasticity [Table 5.4.2 and 6.4.2]</td>
</tr>
<tr>
<td>KST</td>
<td>1.0</td>
<td>Service Condition Factor for Tension, Applicable for tension parallel to grain [Table 5.4.2 and 6.4.2]</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>KSV</td>
<td>1.0</td>
<td>Service Condition Factor for Shear, Applicable for longitudinal shear [Table 5.4.2 and 6.4.2]</td>
</tr>
<tr>
<td>KX</td>
<td>1.0</td>
<td>K value for flexural torsional buckling</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local Y-axis, usually minor axis</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local Z-axis, usually major axis</td>
</tr>
<tr>
<td>KZB</td>
<td>1.0</td>
<td>Size Factor for Bending, Applicable for bending [Clause.5.4.5 and Table 5.4.5]</td>
</tr>
<tr>
<td>KZCP</td>
<td>1.0</td>
<td>Size Factor for Compression, Applicable for compression perpendicular to grain [Clause 5.4.5 and Table 5.4.5]</td>
</tr>
<tr>
<td>KZT</td>
<td>1.0</td>
<td>Size Factor for Tension, Applicable for tension parallel to grain [Clause 5.4.5 and Table 5.4.5]</td>
</tr>
<tr>
<td>KZV</td>
<td>1.0</td>
<td>Size Factor for Shear [Clause 5.4.5 and Table 5.4.5]</td>
</tr>
<tr>
<td>KSCP</td>
<td>1.0</td>
<td>Service Condition Factor for Compression, Applicable for compression perpendicular to grain [Clause 5.4.2 and Table 6.4.2]</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td>Treatment Factor [Clause 5.4.3/6.4.4]</td>
</tr>
<tr>
<td>KZC</td>
<td>1.0</td>
<td>Size Factor for Compression, Applicable for compression parallel to grain [Clause 5.4.5 and Table 5.4.5]</td>
</tr>
</tbody>
</table>
### Design

**D. Design Codes**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LX</td>
<td>Member length</td>
<td>Length for flexural torsional buckling</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Length in local Y axis for slenderness value KL/r</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Length in local Z axis for slenderness value KL/r</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible Ratio of Actual to Allowable Value</td>
</tr>
</tbody>
</table>

**D4.D.7 Code Checking**

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per the CSA086-01 requirements.

Code checking is done using forces and moments at specified sections of the members. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed.

Refer to [D1.G.6 Member Design Capabilities](#) (on page 1538) for general information on Code Checking. Refer to [TR.52.2 Code Checking Specification](#) (on page 2858) for details the specification of the Code Checking command.

```plaintext
PARAMETER
CODE TIMBER CAN
KD 0.99 ALL
KH 0.99 ALL
K_T 0.99 ALL
KSB 0.99 ALL
KSV 0.99 ALL
KSC 0.99 ALL
KSE 0.99 ALL
KST 0.99 ALL
KZB 0.99 ALL
KZV 0.99 ALL
KZT 0.99 ALL
KZCP 0.99 ALL
K_ZC 0.99 ALL
CV 0.99 ALL
KN 0.99 ALL
K_SCPO.99 ALL
CHIX 0.99 ALL
RATIO 0.99 ALL
CHECK CODE ALL
FINISH
```
D4.D.8 Member Selection

Member selection based CSA086-2001 is not available.

D4.D.9 Tabulated Results of Timber Design

Results of code checking and member selection are presented in a tabular format. The term CRITICAL COND refers to the section of the CSA086-01 specification, which governed the design.

- **Pu**: Actual Load in Compression
- **Tu**: Actual Load in Tension
- **Muy**: Ultimate moment in y direction
- **Muz**: Ultimate moment in z direction
- **V**: Ultimate shear force
- **SLENDERNESS_Y**: Actual Slenderness ratio in y direction
- **SLENDERNESS_Z**: Actual Slenderness ratio in z direction
- **PY**: Factored Compressive capacity in y direction
- **PZ**: Factored Compressive capacity in z direction
- **T**: Factored tensile capacity
- **MY**: Factored moment of resistance in y direction
- **MZ**: Factored moment of resistance in z direction
- **V**: Factored shear resistance
- **SLENDERNESS**: Allowable slenderness ratio

D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14

STAAD.Pro is capable of performing steel design based on the Canadian codes CAN/CSA-S16-09 and CAN/CSA-S16-14 Limit States Design of Steel Structures.

D4.E.1 General Comments

The design of structural steel members in accordance with the specification CAN/CSA S16-09 and S16-14 Limit States Design of Steel Structures is can be used in STAAD.Pro This code supercedes the previous edition of the code CAN/CSA – S16-01.

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-states are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the probability of limits being surpassed is acceptably low.

In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths,
desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD.Pro implementation of CAN/CSA-S16-09 and S16-14. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D4.E.2 Analysis Methodology

The elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the specified primary and combination loading condition. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D4.E.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to the STAAD.Pro Technical Reference Manual.

D4.E.4 Built-in Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Almost all Canadian steel sections are available for input. A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Refer to D4.B.4 Built-in Steel Section Library (on page 1701) for additional details.

D4.E.5 Section Classification

Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11 and Table 1 of CAN/CSA-S16-09 or S16-14). The design procedures are different depending on the section class. STAAD.Pro determines the section classification for the standard shapes and user specified shapes.

D4.E.6 Member Resistances

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes
into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

\[ \phi = 0.9 \text{ and } \phi_u = 0.75 \]

**D4.E.6.1 Members Subject to Axial Forces**

**Axial Tension**

The criteria governing the capacity of tension members are based on two limit states: resistance due to yielding and resistance due to rupture. The resistance due to rupture depends on effective net section area. You may specify the net section area through the NSF design parameter. STAAD.Pro calculates the tension capacity of a member based on these two limits states per Cl.13.2 of CAN/CSA-S16-09. Design parameters FYLD, FU, and NSF (Refer to **D4.E.7 Design Parameters** (on page 1736)) are applicable for these calculations

i. Yielding, per Cl. 13.2(a)

\[ T_r = \phi A_g F_y \]

ii. Rupture, per Cl. 13.2 (b)

\[ T_r = \phi_u A_ne F_u \]

**Note:** Pin connection equations in S16-14 are not checked by the program.

**Axial Compression**

The compressive resistance of columns is determined based on Clause 13.3 of the code. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (KL/r ratios). The effective length for the calculation of compression resistance may be provided through the use of the parameters KT, KY, KZ, LT, LY, and LZ (Refer to **D4.E.7 Design Parameters** (on page 1736)). Some of the aspects of the axial compression capacity calculations are:

**I.** For doubly symmetric sections meeting the requirement of Table 1, resistance is:

Resistance due to Major axis buckling per Cl. 13.3.1.

Resistance due to Minor axis buckling per Cl. 13.3.1

\[ C_r = \phi AF_y (1 + \lambda^{2n})^{-1/n} \]

where

\[ n = \begin{cases} 1.34 \text{ for hot-rolled, fabricated structural sections and hollow structural sections manufactured in accordance with CSA G40.20, Class C (cold-formed non-stress-relieved)} \\ 2.24 \text{ for doubly symmetric welded three-plate members with flange edges oxy-flame-cut and hollow structural sections manufactured in accordance with CSA G40.20, Class H (hot-formed or cold-formed stress-relieved)} \end{cases} \]

Design parameters NCR and STP are used to evaluate the value of n for a member.

\[ \lambda = \sqrt{\frac{F_y}{F_e}} \]

\[ F_e = \frac{n^2 E}{(kL/r)^2} \]
II. For any other section not covered under Cl. 13.3.1, the factored compressive resistance, $C_r$, is computed using the expression given in Cl. 13.3.1 with a value of $n = 1.34$ and the value of $F_e$ taken as follows:

i. For doubly symmetric sections and axisymmetric sections, the least of $F_{ex}$, $F_{ey}$, and $F_{ez}$.

ii. For singly symmetric sections with the Y axis taken as the axis of symmetry, the lesser of $F_{ex}$ and $F_{eyz}$ where

\[
F_{eyz} = F_ey + F_ez \left(\frac{1}{r_y} - \frac{4F_eyF_eq}{(F_ey + F_eq)^2}\right)
\]

\[
F_{ex} = \frac{n^2E}{r_x}
\]

\[
F_{ey} = \frac{n^2E}{r_y}
\]

\[
F_{ez} = \frac{n^2E}{C_w} \left(\frac{1}{(K_{Lz})^2} + GJ\right) \frac{1}{Ar_0^2}
\]

\[
x_{0,y_0} = \text{shear center}
\]

\[
r_0^2 = x_0^2 + y_0^2 + r_x^2 + r_y^2
\]

\[
\Omega = 1 - \left(\frac{x_0^2 + y_0^2}{r_0^2}\right)
\]

iii. For asymmetric sections the smallest root of:

\[
\left(F_e - F_{ex}\right)\left(F_e - F_{ey}\right)\left(F_e - F_{ez}\right) - F_e^2 \left(F_e - F_{ey}\right)\left(\frac{x_0}{r_0}\right)^2 - F_e^2 \left(F_e - F_{ex}\right)\left(\frac{y_0}{r_0}\right)^2 = 0
\]

III. For Class 4 member subjected to axial compression, the factored compressive resistance is:

\[
C_r = \phi A_e F_y (1 - \lambda^2n)^{-1/n}
\]

$A_e$ is calculated using reduced element widths meeting the maximum width to thickness ratio specified in Table 1.

Effective width required for the calculation of effective area $A_e$, for different section shapes are as follows.

- For flanges of I-section, T-section and channel section and legs of angle section

  \[
b_e = 200t \sqrt{F_y}
\]

- For stem of T-section

  \[
b_e = 340t \sqrt{F_y}
\]

- For flanges of HSS rectangular or Tube sections

  \[
b_e = 670t \sqrt{F_y}
\]

- For circular HSS or Pipe section

  \[
D = 23,000t/(F_y)
\]

CAN/CSA S16-14 Edition Checks

In S16-14, the flanges of I-shaped sections about major and minor axis is considered by the program.

In S16-14, legs of angles in axial compression are also checked for the following limit state:
The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified in STAAD.Pro through the UNT and UNB parameters (Refer to D4.E.7 Design Parameters (on page 1736)). The LAT parameter is used to specify if the member is laterally supported against lateral torsional buckling.

I. The factored moment resistance, Mr, developed by a member subjected to uniaxial bending moments about a principal axis where effectively continuous lateral support is provided to the compression flange or where the member has no tendency to buckle laterally, is calculated as:

i. For Class 1 and Class 2 sections (Cl. 13.5(a)):

\[ M_r = \phi \cdot Z \cdot F_y = \phi \cdot M_p \]

ii. For Class 3 sections (Cl. 13.5(b)):

\[ M_r = \phi \cdot S \cdot F_y = \phi \cdot M_y \]

iii. For Class 4 sections (Cl. 13.5(c)):

\[ M_r = \phi \cdot S_e \cdot F_y \]

where

\[ S_e = \]  the effective section modulus determined using an effective flange width, \( b_e \), of \( 670t / \sqrt{F_y} \) for flanges along two edges parallel to the direction of stress and an effective flange width, \( b_e \), of \( 200t / \sqrt{F_y} \) for flanges supported along one edge parallel to the direction of stress. For flange supported along one edge, \( b_e I_e/t \) shall not exceed 60.

II. For laterally unsupported members, flexural resistance is calculated as follows:

i. For doubly symmetric Class 1 and Class 2 sections (Cl 13.6(a)):

\[ M_r = \begin{cases} 
\phi M_u & \text{when } M_u \leq 0.67M_p \\
1.15\phi M_p \left(1 - \frac{0.28M_u}{M_p}\right) & \text{when } M_u > 0.67M_p
\end{cases} \]

where

\[ M_u = \frac{\omega_2^2}{L} \left( EI_y GJ + \left(\frac{\pi E}{L}\right)^2 I_y C_w \right) \]

\[ \omega_2 = 1.75 + 1.05\kappa + 0.3\kappa^2 \leq 2.5 \]

\[ \kappa = \]  ratio of smaller factored moment to the larger factored moment at opposite ends of the unbraced length (positive for double curvature and negative for single curvature).

\[ \phi = 1.75 + 1.05\kappa + 0.3\kappa^2 \leq 2.5 \]

\[ \kappa = \]  ratio of smaller factored moment to the larger factored moment at opposite ends of the unbraced length (positive for double curvature and negative for single curvature).

Note: The value for \( \omega_2 \) can be specified using the CB parameter. Otherwise, it is calculated as indicated here.

ii. For doubly symmetric Class 3 and Class 4 sections –except closed square and circular sections– and for channels:
\[ M_r = \begin{cases} 
\phi M_u & \text{when } M_u \leq 0.67M_y \\
1.15\phi M_y \left(1 - \frac{0.28M_y}{M_u}\right) \leq \phi M_p & \text{when } M_u > 0.67M_y 
\end{cases} \]

but not greater than \( \phi M_y \) for Class 3 sections and the value specified in Cl.13.5(c)(iii) for Class 4 sections.

iii. For singly symmetric (monosymmetric) Class 1, Class 2, or Class 3 sections and T-shape sections, lateral torsional buckling strength shall be checked separately for each flange under compression under factored loads at any point along its unbraced length:

- when \( M_u > M_{yr} \):
  \[ M_r = \phi \left[M_p - \left(M_p - M_{yr}\right)\left(\frac{L - L_u}{L_{yr} - L_u}\right)\right] \leq \phi M_p \]
  where
  - \( M_{yr} = 0.7S_x F_y \), with \( S_x \) taken as the smaller of the two potential values
  - \( L_{yr} \) = length \( L \) obtained by setting \( M_u = M_{yr} \)
  - \( L_u = 1.1\sqrt{E/F_y} = \frac{490r_t}{\sqrt{F_y}} \)
  - \( r_t = \frac{b_c}{\sqrt{12(1 + \frac{h_c w}{3b_c t_c})}} \)
  - \( h_c \) = depth of the web in compression
  - \( b_c \) = width of the compression flange
  - \( t_c \) = thickness of the compression flange

- when \( M \leq M_{yr} \):
  \[ M_r = \phi M_u \]
  where
  - \( M_u \) = the critical elastic moment of the unbraced section = \( \frac{\omega_3\pi^2EI_y}{2L} \left[\beta_x + \sqrt{\beta_x^2 + 4\left(\frac{GJ}{\mu^2EI_y} + \frac{C_w}{F_y}\right)}\right] \)
  - \( \beta_x \) = asymmetry parameter for singly symmetric beam = \( 0.9\left(d - t\right)\left(\frac{2I_{yc}}{I_y} - 1\right)\left[1 - \left(\frac{I_y}{I_x}\right)^2\right] \)
  - \( I_{yc} \) = moment of inertia of the compression flange about the y-axis
  - \( I_{yt} \) = moment of inertia of the tension flange about the y-axis

when singly symmetric beams are in single curvature,
\( \omega_3 = \omega_2 \) for beams with two flanges, \( = 1.0 \) for T-sections

in all other cases,
\( \omega_3 = \omega_2(0.5 + 2(I_{yc}/I_y)^2) f, \text{ but } \leq 2.0 \) for T-Sections

D4.E.6.3 Members Subject to Combined Forces

For each of the following interaction equations, the value of the RATIO parameter is used in lieu of 1.0 when it is specified (Refer to D4.E.7 Design Parameters (on page 1736)).
Axial compression and bending

The member strength and stability for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. In these equations, the additional bending caused by the action of the axial load is accounted for by using amplification factors (Cl. 13.8). \( \omega_{1x} \) and \( \omega_{1z} \) are calculated as per Cl. 13.8.5 or as specified in the CYM and CMZ design parameters, respectively.

I. For Class 1 and Class 2 sections of I-shaped members (Cl. 13.8.2):

\[
\frac{C_f}{C_r} + \frac{0.85 U_{1x} M_{fx}}{M_{rx}} + \frac{\beta U_{1y} M_{fy}}{M_{ry}} \leq 1.0
\]

where

\[ C_f, M_f = \] the maximum load effects, including stability, as specified in Cl. 8.4.
\[ \beta = 0.6 + 0.4\lambda_y \leq 0.85 \]

The capacity of the member is investigated for the following:

a. Cross sectional strength with \( \beta = 0.6 \), where
   i. \( C_r \) as specified in Cl. 13.3 with \( \lambda = 0 \)
   ii. \( M_r \) as specified in Cl. 13.5
   iii. \( U_{1x} \) and \( U_{1y} \) as specified in Cl. 13.8.4 but not less than 1.0. Design parameters SSY and SSZ are used to evaluate these coefficients.

b. Overall member strength, where
   i. \( C_r \) as specified in Cl. 13.3 with \( K = 1 \), except for uniaxial bending, in which case \( C_r \) is based on the axis of bending
   ii. \( M_r \) as specified in Cl. 13.5
   iii. \( U_{1x} \) and \( U_{1y} \) are taken as 1.0 for members in an unbraced frame, and as specified in Cl. 13.8.4 for members in a braced frame. Design parameters SSY and SSZ are used to evaluate these coefficients.

c. Lateral torsional buckling strength, when applicable, where
   i. \( C_r \) as specified in Cl. 13.3
   ii. \( M_{rx} \) as specified in Cl. 13.6
   iii. \( M_{ry} \) as specified in Cl. 13.5
   iv. \( U_{1x} \) and \( U_{1y} \) are taken as 1.0 for members in an unbraced frame, and as specified in Cl. 13.8.4 for members in a braced frame (where \( U_{1x} \) is not less than 1.0). Design parameters SSY and SSZ are used to evaluate these coefficients.

II. For all other cases (Cl. 13.8.3):

\[
\frac{C_f}{C_r} + \frac{U_{1x} M_{fx}}{M_{rx}} + \frac{U_{1y} M_{fy}}{M_{ry}} \leq 1.0
\]

The capacity of the member is investigated for the following per Cl. 13.8.2:

a. Cross sectional strength
b. Overall member strength
c. Lateral torsional buckling strength,

Axial tension and bending

Members subjected to axial tension and bending must satisfy the following equation (Cl. 13.9.1):

\[
\frac{T_f}{T_r} + \frac{M_f}{M_r} \leq 1.0
\]
where

\[ M_r = \text{the moment resistance as specified in Cl. 13.5.} \]

Additionally, the following equations must be satisfied for laterally unsupported members (Cl. 13.9.2):

\[
\frac{M_f}{M_r} - \frac{T_f Z}{M_r A} \leq 1.0 \text{ for Class 1 and Class 2 sections}
\]

\[
\frac{M_f}{M_r} - \frac{T_f S}{M_r A} \leq 1.0 \text{ for Class 3 and Class 4 sections}
\]

where

\[ M_r = \text{the moment resistance as specified in Cl. 13.6.} \]

**Biaxial Bending**

For bending about both axis, the following equation must be satisfied (Cl. 13.8):

\[
\frac{M_{fx}}{M_{rx}} + \frac{M_{fy}}{M_{ry}} \leq 1.0
\]

**Shear and Bending**

To resist the combined effects of shear and bending, all of the following equations must be satisfied (Cl. 14.6):

\[
0.727 \frac{M_f}{M_r} + 0.455 \frac{V_f}{V_r} \leq 1.0
\]

\[
\frac{M_f}{M_r} \leq 1.0
\]

\[
\frac{V_f}{V_r} \leq 1.0
\]

where

\[ M_r = \text{the value determined in accordance with Cl. 13.5 of Cl 13.6 as applicable} \]

\[ V_r = \text{the value determined in accordance with Cl. 13.4} \]

**D4.E.6.4 Shear**

Factored shear resistance, \( V_r \), developed by the web of flexural member is calculated as:

\[ V_r = \phi A_w F_s \]

where

\[ A_w = \text{shear area} \]

\( F_s \) is evaluated as:

**I. For unstiffened webs (Cl. 13.4.1.1(a)):**

i. when \( \frac{h}{w} \leq \frac{1.014}{\sqrt{F_y}} \), \( F_s = 0.66F_y \)

ii. when \( \frac{1.014}{\sqrt{F_y}} < \frac{h}{w} \leq \frac{1.435}{\sqrt{F_y}} \), \( F_s = \frac{670\sqrt{F_y}}{(h / w)} \)

iii. when \( \frac{h}{w} > \frac{1.436}{\sqrt{F_y}} \), \( F_s = \frac{961}{(h / w)^2} \)

---

**Notes:**

- \( M_f \) and \( T_f \) are factored moments and forces, respectively.
- \( Z \) and \( S \) are the section modulus and section plastic modulus, respectively.
- \( A \) and \( A_w \) are the area of the section and shear area, respectively.
- \( F_y \) is the yield strength of the material.
II. For stiffened webs (i.e., when the STIFF parameter is specified) (Cl. 13.4.1.1(b)):

i. when \( \frac{h}{w} \leq 439 \sqrt{\frac{k_v}{F_y}}, F_s = 0.66F_y \)

ii. when \( 439 \sqrt{\frac{k_v}{F_y}} < \frac{h}{w} \leq 502 \sqrt{\frac{k_v}{F_y}}, F_s = F_{cri} \)

iii. when \( 502 \sqrt{\frac{k_v}{F_y}} < \frac{h}{w} \leq 621 \sqrt{\frac{k_v}{F_y}}, F_s = F_{cri} + k_a(0.50F_y \cdot 0.866F_{cri}) \)

iv. when \( 621 \sqrt{\frac{k_v}{F_y}} < \frac{h}{w} \), \( F_s = F_{cre} + k_a(0.50F_y \cdot 0.866F_{cre}) \)

where

\( A_e = \) shear buckling coefficient:

i. when \( a/h < 1 \), \( k_v = 4 + \frac{5.34}{(a/h)^2} \)

ii. when \( a/h \geq 1 \), \( k_v = 5.34 + \frac{4}{(a/h)^2} \)

\( a/h = \) stiffener aspect ratio (i.e., ratio of the distance between stiffeners to web depth)

\( F_{cri} = 290 \sqrt{\frac{F_y k_v}{(h/w)}} \)

\( k_a = \) aspect coefficient = \( \frac{1}{\sqrt{1 + (a/h)^2}} \)

\( F_{cre} = \frac{180,000k_v}{(h/w)^2} \)

For tubular members, the shear resistance, \( V_r \), is calculated as:

\[ V_r = 0.66\phi (A_e/2)F_y \]

where

\( A_e = \) the cross-sectional area of the tubular member

D4.E.7 Design Parameters

The design parameters outlined in the following table may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application’s specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.
Table 140: Canadian Steel Design CSA-S16-09/14 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td></td>
<td>Must be specified as CANADIAN for S16-14. Must be specified as CANADIAN 2009 for S16-09. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>Used to specify locations along member length considered for design: 0.0 = design only for end moments and those at locations specified by a SECTION command. 1.0 = Perform design for moments at twelfth points along the beam.</td>
</tr>
<tr>
<td>CB</td>
<td>0.0</td>
<td>Value of $\omega_2$ (code Cl 13.6) to be used for calculations. $\omega_2$ is calculated internally if CB is 0.0 or not provided.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CMY</td>
<td>( \omega_1 ) calculated as per clause number 13.8.5 (a)</td>
<td>( \omega_1 ) in local Y direction for the member. The program calculated value of ( \omega_1 ) depends on whether the member is subjected to any transverse load or not:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- ( \omega_1 = 0.6 - 0.4\kappa \geq 0.4 ) if the member is not subjected to transverse loads between supports. Refer Clause 13.8.5 (a)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- ( \omega_1 = 1.0 ) if the member is subjected to distributed loads or a series of point loads between supports. Refer Clause 13.8.5 (b)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- ( \omega_1 = 0.85 ) if the member is subjected to a concentrated load or moment between supports. Refer Clause 13.8.5 (c)</td>
</tr>
<tr>
<td>CMZ</td>
<td>( \omega_1 ) calculated as per clause number 13.8.5 (a)</td>
<td>( \omega_1 ) in local Z direction for the member. The program calculated value of ( \omega_1 ) depends on whether the member is subjected to any transverse load or not:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- ( \omega_1 = 0.6 - 0.4\kappa \geq 0.4 ) if the member is not subjected to transverse loads between supports. Refer Clause 13.8.5 (a)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- ( \omega_1 = 1.0 ) if the member is subjected to distributed loads or a series of point loads between supports. Refer Clause 13.8.5 (b)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- ( \omega_1 = 0.85 ) if the member is subjected to a concentrated load or moment between supports. Refer Clause 13.8.5 (c)</td>
</tr>
</tbody>
</table>

STAAD.Pro will calculate \( \omega_1 \) based on the formula specified in 13.8.5 (a) unless you directly specify a value. You may specify any value between 0.4 to 1 for \( \omega_1 \).
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPSACING</td>
<td>0.0</td>
<td>Spacing between connectors of built-up members required for slenderness ratio calculation per Cl. 19.1.4.</td>
</tr>
<tr>
<td>DFF</td>
<td>None(Mandatory for deflection check)</td>
<td>“Deflection Length”/Maximum Allowable local deflection.</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting start point for calculation of “Deflection Length”</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of “Deflection Length”</td>
</tr>
<tr>
<td>DMAX</td>
<td>45.0 in.</td>
<td>Maximum allowable depth (Applicable for member selection)</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in.</td>
<td>Minimum required depth (Applicable for member selection)</td>
</tr>
</tbody>
</table>
| FLX            | 0             | Parameter for specifying the flexural-torsional restraint condition.  
|                |               | 0 = Flexural-torsional restraint is not provided  
|                |               | 1= Flexural-torsional restraint is provided along the length. |
| FU             | 345.0 MPa     | Ultimate strength of steel. |
| FYLD           | 300.0 MPa     | Yield strength of steel. |
| KT             | 1.0           | K value for flexural torsional buckling. |
| KY             | 1.0           | K value for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio. |
| KZ             | 1.0           | K value for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio. |
| LAT            | 0             | Specify lateral support conditions:  
|                |               | 0 = Beam is laterally unsupported.  
<p>|                |               | 1 = Beam is laterally supported. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LEG</td>
<td>0</td>
<td>This parameter is meant for plain angles (clause 13.3.3.2).&lt;br&gt;0 = The angle is connected by the longer leg.&lt;br&gt;1 = The angle is connected by the shorter leg.</td>
</tr>
<tr>
<td>LT</td>
<td>Member Length</td>
<td>Length for flexural torsional buckling.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Length for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.</td>
</tr>
<tr>
<td>MAIN</td>
<td>200</td>
<td>Allowable slenderness limit for compression members.</td>
</tr>
<tr>
<td>NCR</td>
<td>0</td>
<td>This parameter sets the n factor to calculate Cr value as per clause number 13.3.1&lt;br&gt;0 = 1.34&lt;br&gt;1 = 2.24</td>
</tr>
</tbody>
</table>

**Notes:**

1. The default value of n is used for all sections.
2. n = 2.24 for Hollow sections (e.g., HSST, HSSP, Pipe Tube), for doubly symmetric built-up (STP = 2) sections (e.g., W, M, S, HP sections), and for Welded Wide Flange (WWF) sections if NCR is set to 1. WWF sections are designed as built up sections.
3. n = 1.34 for doubly symmetric hot rolled sections and for single symmetric hot rolled or built-up sections (e.g., Channel, Tee, Angle) regardless of the NCR parameter value.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>PROFILE</td>
<td>-</td>
<td>Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual load effect to the design strength.</td>
</tr>
</tbody>
</table>
| SNUG           | 1             | Specify type of connection for the built-up members (Refer to Cl. 19.1.4):  
|                |               | 0 = Welded or pretensioned bolts.  
|                |               | 1 = Bolted snug-tight. |
| SSY            | 0             | 0 = Sway or Unbraced along local Y axis  
|                |               | 1 = Braced along local Y axis  
|                |               | This parameter is required to choose the proper value of $U_{1y}$ which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength. Refer to Axial compression and bending (on page 1734) for details. |
| SSZ            | 0             | 0 = Sway or Unbraced along local Z axis  
|                |               | 1 = Braced along local Z axis  
<p>|                |               | This parameter is required to choose the proper value of $U_{1z}$ which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength. Refer to Axial compression and bending (on page 1734) for details. |
| STIFF          | Member length of depth of beam, whichever is lesser. | Spacing of traverse stiffeners. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| STP            | 1             | 1.0 = Rolled section  
                2.0 = Welded built-up section |
| TRACK          | 0.0           | Design output:  
                0.0 = Report only minimum design results.  
                1.0 = Report design strengths also.  
                2.0 = Provide full details of design. |
| UNB            | Member Length | Unsupported length in bending compression of the bottom flange for calculating moment resistance. |
| UNT            | Member Length | Unsupported length in bending compression of the top flange for calculating moment resistance. |

**D5. European Codes**

**D5.A. European Codes - Concrete Design Per DD ENV 1992**

*Note:* This code has been removed from the batch design. To perform design to the current Eurocode 2 design code, please use the interactive design in the [D. European Concrete Design per Eurocode EC2: 2004](#) (on page 1044).

**D5.B. European Codes - Steel Design per Eurocode 3 [DD ENV 1993-1-1:1992]**


*Note:* The DD ENV 1993-1-1:1992 code has now been officially superseded by EN 1993-1-1:2005. Hence releases of STAAD.Pro subsequent to version SS3 (20.07.08.xx) will not support this design code. The SS3 build will perform member design to this code for legacy files but has this code removed from the design codes list in the GUI. Users are advised to use the EN 1993-1-1:2005 version for Eurocode 3 design.

**Tip:** Design per EC3 DD ENV 1993-1-1:1992 is also available in the Steel Design mode in the Graphical User Interface.

**D5.B.1 General Description**

1. Selecting the applicable load cases to be considered in the design process.
2. Providing appropriate “Parameter” values if different from the default values.
3. Specify whether to perform code-checking and/or member selection.

These operations can be repeated by the user any number of times depending on the design requirements. The “Parameters” referred to above provide the user with the ability to allocate specific design properties to individual members or member groups considered in the design operation.

**D5.B.1.1 Eurocode 3 DD ENV 1993-1-1:1992 (EC3 DD)**

The DD ENV version of Eurocode 3, *Design of steel structures, Part 1.1 General rules and rules for buildings* (EC3 DD) provides design rules applicable to structural steel used in buildings and civil engineering works. It is based on the ultimate limit states philosophy that is common to modern standards. The objective of this method of design is to ensure that possibility of failure is reduced to a negligible level. This is achieved through application of safety factors to both the applied loads and the material properties.

The code also provides guidelines on the global methods of analysis to be used for calculating internal member forces and moments. STAAD uses the elastic method of analysis which may be used in all cases. Also there are three types of framing referred to in EC3. These are “Simple”, “Continuous”, and “Semi-continuous” which reflect the ability of the joints to developing moments under a specific loading condition. In STAAD only “Simple” and “Continuous” joint types can be assumed when carrying out global analysis.

**D5.B.1.2 National Application Documents**

Various authorities of the CEN member countries have prepared National Application Documents to be used with EC3. These documents provide alternative factors for loads and may also provide supplements to the rules in EC3.

The current version of EC3 DD implemented in STAAD adheres to the factors and rules provided in DD ENV 1993-1-1:1992 and has not been modified by any National Application Document.

**Note:** National Annex documents are available for EC3 BS EN 1993-1-1:2005. Refer to [D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005]](on page 1754)

**Axes convention in STAAD.Pro and Eurocode 3**

By default, STAAD.Pro defines the major axis of the cross-section as Z-Z and the minor axis as Y-Y. A special case where Z-Z is the minor axis and Y-Y is the major axis is available if the SET Z UP command is used and is discussed in [TR.5 Set Command Specification](on page 2413). The longitudinal axis of the member is defined as X and joins the start joint of the member to the end with the same positive direction.

Eurocode 3, however, defines the principal cross-section axes in reverse to that of STAAD.Pro, but the longitudinal axis is defined in the same way. Both of these axes definitions follow the orthogonal right hand rule.

Bear this difference in mind when examining the code-check output from STAAD.Pro.
D5.B.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations.

D5.B.3 Material Properties and Load Factors

The characteristic yield strength of steel used in EC3 DD design is based on table 3.1 of the code. Design resistances are obtained by dividing the characteristic yield strength by the material partial safety factor $\Gamma_m$. The magnitude of $\Gamma_m$ in STAAD.Pro is 1.1 which is applicable to all section types. A separate safety factor parameter named GB1 is used to check the resistance of a member to buckling and also has a default value of 1.1.

Material coefficients for steel in STAAD.Pro take the following default values unless replaced by user’s numerical values provided in the input file.

- Modulus of Elasticity, $E = 205000 \text{ N/mm}^2$
- Shear Modulus, $G = E/2(1+\nu)$
- Poisson’s Ratio, $\nu = 0.3$
- Unit weight, $\Gamma = 76.8 \text{ KN/m}^3$

The magnitude of design loads is dependent on $\Gamma_f$, the partial safety factor for the action under consideration. In STAAD.Pro you are allowed total control in providing applicable values for the factors and their use in various load combinations.

D5.B.4 Section Classification

The occurrence of local buckling of the compression elements of a cross-section prevents the development of full section capacity. It is therefore imperative to establish this possibility prior to determining the section capacities. Cross sections are classified in accordance with their geometrical properties and the stress pattern on the compression elements. For each load case considered in the design process, STAAD determines the section class and calculates the capacities accordingly.

The EC3 DD design module in STAAD can design members with all section profiles that are of Class 1 2 or 3 as defined in section 5.3.2 of the code. However, the design of members that have a “Class 4” section profile are limited to WIDE FLANGE, TEE, SINGLE CHANNEL, SINGLE ANGLE, and RECTANGULAR HOLLOW SECTIONS. Also built-up user sections that are class 4 sections are not dealt with in the current version of EC3 design in STAAD.Pro.
Laced and battened members are not considered in the current version of EC3 DD design module in STAAD.Pro.

D5.B.5 Member Design

D5.B.5.1 Design of Beams as per DD ENV 1993-1-1:1992

EC3 DD design in STAAD.Pro considers members that are primarily in bending and/or shear as beams and performs cross section and member capacity checks in accordance with the code. The main requirement for a beam is to have sufficient cross-section resistance to the applied bending moment and shear force. The possibility of lateral-torsional buckling is also taken into consideration when the full length of the member has not been laterally restrained.

The bending capacity is primarily a function of the section type and the material yield strength and is determined according to Cl. 5.4.5 of the code. The shear capacity and the corresponding shear checks are done as per section 5.4.6 of the code.

There are four classes of cross-sections defined in EC3. Class 1 and 2 sections can both attain full capacity with the exception that the class 2 sections cannot sustain sufficient rotation required for plastic analysis of the model. Hence the full plastic section modulus is used in the design calculations. Class 3 sections, due to local buckling, cannot develop plastic moment capacity and the yield stress is limited to the extreme compression fibre of the section. The elastic section modulus is used to determine the moment capacity for class 3 sections. Class 4 sections do suffer from local buckling and explicit allowance must be made for the reduction in section properties before the moment capacity can be determined. Further, because of interaction between shear force and bending moment, the moment resistance of the cross-section may be reduced. This, however, does not occur unless the value of applied shear forces exceeds 50% of the plastic shear capacity of the section. In such cases the web is assumed to resist the applied shear force as well as contributing towards the moment resistance of the cross-section.

As mentioned in the previous section, the design of class 4 sections is limited to WIDE FLANGE, TEE, SINGLE CHANNEL, SINGLE ANGLE, and RECTANGULAR HOLLOW SECTIONS. The effective section properties are worked out as described in Cl. 5.3.5 of the code.

Beams are also checked for lateral-torsional buckling according to section 5.5.2 of the code. The buckling capacity is dependent on the section type as well as the unrestrained length, restraint conditions and type of applied loading. The lateral torsional buckling checks involves the calculation of the “Elastic critical moment”, Mcr, which is calculated in STAAD as per the method given in Annex F of the code.

In the presence of a shear force, beams are also checked for shear as per section 5.4.6 of the code. In cases where the members are subject to combined bending and shear, the combined bending and shear checks are done in STAAD as per clause 5.4.7 of the code.

D5.B.5.2 Design of Axially Loaded Members

The design of members subject to tension loads alone are performed as per Cl 5.4.3 of the code. The tension capacity is calculated based on yield strength, material factor \( \Gamma_m \) and cross-sectional area of the member with possible reduction due to bolt holes. When bolt holes need to be considered in the capacity calculations the value used for \( \Gamma_m \) is 1.2 and the yield strength is replaced with the ultimate tensile strength of the material. The tension capacity is then taken as the smaller of the full section capacity and the reduced section capacity as stated above.

The design of members subject to axial compression loads alone are performed as per Cl 5.4.4 of the code. For members with class 1 2 or 3 section profiles, the full section area is considered in calculating the section capacity. However in case of class 4 sections, the "effective cross-section" is considered to calculate the
compressive strength. Also any additional moments induced in the section due to the shift of the centroidal axis of the effective section will also be taken into account as per clause 5.4.8.3 of the code. The effective section properties for class 4 sections will be worked out as given in Cl.5.3.5 of the code.

In addition to the cross section checks, buckling resistance will also be checked for such members. This is often the critical case as the buckling strength of the member is influenced by a number of factors including the section type and the unbraced length of the member. The buckling capacity is calculated as per Cl. 5.5 of the code.

DD ENV 1993-1-1:1992 does not specifically deal with single angle, double angles, double channels or Tee sections and does give a method to work out the slenderness of such members. In these cases, the EC3 DD design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 and table 25 of BS 5950-1:2000 are used in the current version of the EC3 DD design module.

Single Angle Sections

Angle sections are un-symmetrical and when using BS 5950:2000 table 25 you must consider four axes: two principal, u-u and v-v and two geometric, a-a and b-b. The effective length for the v-v axis, \( L_{vv} \), is taken as the \( L_{vv} \) parameter or \( L_Y \cdot K_Y \), if not specified. The a-a and b-b axes are determined by which leg of the angle is fixed by the connection and should be specified using the \( \text{LEG} \) parameter, see section 5B.6 for more information on the \( \text{LEG} \) parameter. The effective length in the a-a axis is taken as \( L_Y \cdot K_Y \) and the effective length in the b-b axis as \( L_Z \cdot K_Z \).

The following diagram shows the axes for angles which have been defined with either an ST or RA specification and is connected by its longer leg (i.e., a-a axis is parallel to the longer leg).

![Diagram showing axes for angles](image)

**D5.B.5.3 Design of members with combined axial load and bending**

The bending resistance of members could be reduced by the presence of a co-existent axial load. This is then checked against the lateral-torsional buckling resistance of the section. The EC3 DD design module in STAAD takes such a scenario into account and performs the necessary checks as per Cl. 5.4.8 of the code. Class 1 and class 2 sections are checked as per cl. 5.4.8.1 and Class 3 and Class 4 sections are checked as per clauses 5.4.8.2.
and 5.4.8.3 respectively. The effective section properties for class 4 sections are worked out as given in Cl. 5.3.5 of the code.

Generally, EC3 requires checking cross-section resistance for local capacity and also checking the overall buckling capacity of the member. In the case of members subject to axial tension and bending, there is provision to take the stabilizing effect of the tension load into consideration. This is achieved by modifying the extreme compression fibre stress and calculating an effective applied moment for the section. The checks are done as per Cl. 5.5.3 of the code. In case of a combined axial compressive load and bending moment, the member will be checked as per the rules in section 5.5.4 of the code.

The presence of large shear force can also reduce the bending resistance of the section under consideration. If the shear load is large enough to cause a reduction in bending resistance, then the reduction due to shear has to be taken into account before calculating the effect of the axial load on the bending resistance of the section. If the member is subject to a combined shear, axial load and bending moment then the section capacity checks will be done as per Cl. 5.4.9 of the code.

As stated in the previous section, DD ENV 1993-1-1:1992 does not specifically deal with single angle, double angles, double channels or Tee sections and does give a method to work out the slenderness of such members. In these cases, the EC3 DD design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 of BS 5950-1:2000 is used in the current version of the EC3 DD design module. Please refer to the note in section 5B.5.2 for St and RA angle specifications.

Please note that laced or battened compression members are not dealt within the current version of EC3 DD design module in STAAD.Pro.

D5.B.6 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Depending on the model being designed, the user may have to change some or all of the parameter default values. Some parameters are unit dependent and when altered, the new setting must be compatible with the active “unit” specification.

The following table lists all the relevant EC3 parameters together with description and default values.

### Table 141: Steel Design Parameters EC3 DD

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>Undefined</td>
<td>You must specify EC3 or EUROPE. Design code to follow. See [TR.48.1 Parameter Specifications](on page 2851).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>BEAM</td>
<td>3</td>
<td>Parameter to control the number of sections to be checked along the length of a beam:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Check sections with end forces only</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Check at location of maximum Mz along beam</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Check sections with end forces and forces at location of BEAM 1.0 check.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. Check at every 1/13th point along the beam and report the maximum</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refer to Note 2 below.</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>Member will be considered as a cantilever type member for deflection checks.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 indicates that member will not be treated as a cantilever member</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 indicates that the member will be treated as a cantilever member</td>
</tr>
<tr>
<td>CMM</td>
<td>1.0</td>
<td>Indicates type of loading on member. Valid values range from 1 to 6.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refer to Table 7B.3 for more information on its use.</td>
</tr>
<tr>
<td>CMN</td>
<td>1.0</td>
<td>Indicates the level of End-Restraint.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = No fixity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5 = Full fixity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.7 = One end free and other end fixed</td>
</tr>
<tr>
<td>DMAX</td>
<td>100.0 cm</td>
<td>Maximum allowable depth for the member.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0</td>
<td>Minimum required depth for the member.</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>Deflection limit</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot;.</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot;.</td>
</tr>
<tr>
<td>FU</td>
<td></td>
<td>Ultimate tensile strength of steel</td>
</tr>
<tr>
<td>GB1</td>
<td>1.1</td>
<td>Partial safety factor used in buckling checks for compression members</td>
</tr>
<tr>
<td>GM0</td>
<td>1.1</td>
<td>Corresponds to the $\Gamma_{m0}$ factor in DD ENV 1993-1-1:1992</td>
</tr>
<tr>
<td>GM1</td>
<td>1.1</td>
<td>Corresponds to the $\Gamma_{m1}$ factor in DD ENV 1993-1-1:1992</td>
</tr>
<tr>
<td>GM2</td>
<td>1.1</td>
<td>Corresponds to the $\Gamma_{m2}$ factor in DD ENV 1993-1-1:1992</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K factor in local y axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K factor in local z axis.</td>
</tr>
<tr>
<td>LEG</td>
<td>0.0</td>
<td>Connection type Refer to Note 1 below.</td>
</tr>
<tr>
<td>LVV</td>
<td>Maximum of Lyy and Lzz (Lyy is a term used by BS5950)</td>
<td>Buckling length for angle about its principle axis</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Compression length in local y axis, Slenderness ratio = (KY)*(LY)/(Ryy)</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Compression length in local z axis, Slenderness ratio = (KZ)*(LZ)/(Rzz)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| PLG            | 0             | (Polish NA only) Perform additional checks per Cl. 6.3.3  
0. Ignore additional PN EN checks  
1. Include additional PN EN checks  
Refer to D5.D.8.7 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz (on page 1830) |
| PY             | Yield Strength | The yield strength default value is set based on the default value of the "SGR" parameter. |
| NSF            | 1.0           | Net tension factor for tension capacity calculation. |
| RATIO          | 1             | Permissible ratio of loading to capacity. |
| SBLT           | 0.0           | Indicates if the section is rolled or built-up.  
0.0 = Rolled  
1.0 = Built-up |
| SGR            | 0.0           | Steel grade as per table 3.1 in EC3.  
0.0 = Fe 360  
1.0 = Fe 430  
2.0 = Fe 510 |
| TRACK          | 0             | Controls the level of detail of output.  
0 = minimum  
1 = intermediate  
2 = maximum  
4 = perform a deflection check  
See note 3 below. |
| UNF            | 1.0           | Unsupported buckling length as a factor of the beam length |
### Parameter Name | Default Value | Description
--- | --- | ---
UNL | Member Length | Unrestraint length of member used in calculating the lateral-torsional resistance moment of the member.
ZIV | 0.8 | Specifies a reduction factor for vectoral effects to be used in axial tension checks [Cl 5.5.3(2)]

### Notes

1. **LEG** – (Ref: Table 25 BS5950)

   The slenderness of single and double angle, channel and tee sections are specified in BS 5950 table 25 depending on the connection provided at the end of the member (Refer to section 5B.5(A).2 on page 1744). To define the appropriate connection, a LEG parameter should be assigned to the member.

   The following table indicates the value of the LEG parameter required to match the BS5950 connection definition:

   **Table 142: LEG Parameter values**

<table>
<thead>
<tr>
<th>Clause</th>
<th>Bold Configuration</th>
<th>Leg</th>
<th>LEG Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.7.10.2 Single Angle</td>
<td>(a) - 2 bolts</td>
<td>short leg</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>long leg</td>
<td>3.0</td>
</tr>
<tr>
<td></td>
<td>(b) - 1 bolts</td>
<td>short leg</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>long leg</td>
<td>2.0</td>
</tr>
<tr>
<td>4.7.10.3 Double Angles</td>
<td>(a) - 2 bolts</td>
<td>short leg</td>
<td>3.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>long leg</td>
<td>7.0</td>
</tr>
<tr>
<td></td>
<td>(b) - 1 bolts</td>
<td>short leg</td>
<td>2.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>long leg</td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>(c) - 2 bolts</td>
<td>long leg</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>short leg</td>
<td>5.0</td>
</tr>
<tr>
<td></td>
<td>(d) - 1 bolts</td>
<td>long leg</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>short leg</td>
<td>4.0</td>
</tr>
<tr>
<td>4.7.10.4 Channels</td>
<td>(a) - 2 or more rows of bolts</td>
<td></td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>(b) - 1 row of bolts</td>
<td></td>
<td>0.0</td>
</tr>
</tbody>
</table>
Clause Bold Configuration Leg LEG Parameter
4.7.10.5 Tee Sections (a) - 2 or more rows of bolts 1.0
(b) - 1 row of bolts 0.0

For single angles, the slenderness is calculated for the geometric axes, a-a and b-b as well as the weak v-v axis. The effective lengths of the geometric axes are defined as:

\[
L_a = K_Y \times K_Y \\
L_b = K_Z \times L_Z
\]

The slenderness calculated for the v-v axis is then used to calculate the compression strength \( p_c \) for the weaker principal axis (z-z for ST angles or y-y for RA specified angles). The maximum slenderness of the a-a and b-b axes is used to calculate the compression strength \( p_c \) for the stronger principal axis.

Alternatively for single angles where the connection is not known or Table 25 is not appropriate, by setting the LEG parameter to 10, slenderness is calculated for the two principal axes y-y and z-z only. The LVV parameter is not used.

For double angles, the LVV parameter is available to comply with note 5 in Table 25. In addition, if using double angles from user tables, ( G.6.3 User-Provided Steel Table (on page 2325) ) an eleventh value, \( r_{vv} \), should be supplied at the end of the ten existing values corresponding to the radius of gyration of the single angle making up the pair.

2. BEAM

Ensure that this parameter is set to either 1 or 2 while performing code checking for members susceptible to Lateral - Torsional Buckling.

Table 143: Values for the CMM Parameter

<table>
<thead>
<tr>
<th>CMM Value</th>
<th>Loading and Support Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td><img src="image1.png" alt="Diagram" /></td>
</tr>
<tr>
<td>2</td>
<td><img src="image2.png" alt="Diagram" /></td>
</tr>
<tr>
<td>3</td>
<td><img src="image3.png" alt="Diagram" /></td>
</tr>
<tr>
<td>4</td>
<td><img src="image4.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>
3. Checking beam deflection

With the TRACK parameter set to 4, the members included in a CHECK CODE command will be checked for the local axis deflection rather than for the stress capacity using the current LOAD LIST.

If both stress capacity and deflection checks are required, then 2 parameter blocks with code checks are required, one with a TRACK 4 command and one with a TRACK 0, 1, or 2, thus:

```
LOAD LIST 1 TO 10
PARAMETER 1
CODE EN 1993
TRACK 2 ALL
CHECK CODE MEMBER 1
***************************
LOAD LIST 100 TO 110
PARAMETER 2
TRACK 4 ALL
DFF 300 MEMB 1
DJ1 1 MEMB 1
DJ2 4 MEMB 1
CODE MEMB 1
```

**Note:** While both sets of code checks will be reported in the output file, only the last code check results are reported in the GUI.

### D5.B.7 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate. The adequacy is checked as per DD ENV 1993-1-1:1992. Code checking is done using the forces and moments at specific sections of the members.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks; the critical condition; the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member of forces in the member where the critical condition occurs).

Code checking can be done with any type of steel section listed in Section 2B.4 (on page 1657) or any of the user defined sections as described in G.6.3 User-Provided Steel Table (on page 2325), with two exceptions; GENERAL and ISECTION. The EC3 DD design module does not consider these sections or PRISMATIC sections in its design process.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.
D5.B.8 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN, which limits the maximum and minimum depth of the members.

Member selection can be performed with all the types of steel sections with the same limitations as defined in section 5B.7(A) Code Checking (on page 1753).

Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table.

Member selection cannot be performed on members whose section properties are input as prismatic or as the limitations specified in section 5.B.7(A).

D5.B.9 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

- **MEMBER** refers to the member number for which the design is performed.
- **TABLE** refers to steel section name, which has been checked against the steel code or has been selected.
- **RESULTS** prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
- **CRITICAL COND** refers to the clause in DD ENV 1993-1-1:1992 code which governs the design.
- **RATIO** prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
- **LOADING** provides the load case number, which governed the design.
- **FX, MY, and MZ** provide the axial force, moment in local Y-axis and the moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.
- **LOCATION** specifies the actual distance from the start of the member to the section where design forces govern.

**Note:** For a TRACK 2 output, the module will also report all the relevant clause checks that have been performed and will also indicate the critical ratio and the load case that caused the critical ratio as well as the corresponding forces that were used for the respective checks. A TRACK 2 output will also include the various design data used for the calculations such as the section modulii, section class, section capacity etc.

D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005]

Note: The implementation of EN1993-1-1:2005 includes the amendments as per CEN corrigenda of February 2006 and April 2009.

D5.C.1 General Description

The main steps in performing a design operation are:

1. Selecting the applicable load cases to be considered in the design process.
2. Providing appropriate “Parameter” values if different from the default values.
3. Specify whether to perform code-checking and/or member selection.

These operations can be repeated by the user any number of times depending on the design requirements. The “Parameters” referred to above provide the user with the ability to allocate specific design properties to individual members or member groups considered in the design operation.


The EN 1993 version of Eurocode 3, Design of steel structures, Part 1.1 General rules and rules for buildings (EN 1993) provides design rules applicable to structural steel used in buildings and civil engineering works. It is based on the ultimate limit states philosophy that is common to modern standards. The objective of this method of design is to ensure that possibility of failure is reduced to a negligible level. This is achieved through application of safety factors to both the applied loads and the material properties.

The code also provides guidelines on the global methods of analysis to be used for calculating internal member forces and moments. STAAD uses the elastic method of analysis which may be used in all cases. Also there are three types of framing referred to in EC3. These are “Simple”, “Continuous”, and “Semi-continuous” which reflect the ability of the joints to developing moments under a specific loading condition. In STAAD only “Simple” and “Continuous” joint types can be assumed when carrying out global analysis.

D5.C.1.2 National Annex Documents

Various authorities of the CEN member countries have prepared National Annex Documents to be used with EC3. These documents provide alternative factors for loads and may also provide supplements to the rules in EC3.

The current version of EC3 (EN 1993) implemented in STAAD adheres to the factors and rules provided in EN 1993-1-1:2005. The current version of STAAD.Pro includes the following National Annexes viz.

b. The Dutch National Annex [NEN-EN 1993-1-1/NB] and
g. Singaporean National Annex [SS EN 1993-1-1:2005]

The choice of a particular National Annex is based on the value of a new NA parameter that is set by the user when specifying the EN 1993 version of Eurocode 3. Refer to D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1792) for a description of the NA parameter.
Axes convention in STAAD.Pro and Eurocode 3

By default, STAAD.Pro defines the major axis of the cross-section as Z-Z and the minor axis as Y-Y. A special case where Z-Z is the minor axis and Y-Y is the major axis is available if the `SET Z UP` command is used and is discussed in TR.5 Set Command Specification (on page 2413). The longitudinal axis of the member is defined as X and joins the start joint of the member to the end with the same positive direction.

Eurocode 3, however, defines the principal cross-section axes in reverse to that of STAAD.Pro, but the longitudinal axis is defined in the same way. Both of these axes definitions follow the orthogonal right hand rule.

Bear this difference in mind when examining the code-check output from STAAD.Pro.

Refer to D5.C.9 Tabulated Results of Steel Design (on page 1790) for an example of how this appears when Y is up (default).

D5.C.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations.

D5.C.3 Material Properties and Load Factors

The characteristic yield strength of steel used in EC3 (EN 1993) design is based on table 3.1 of the code. Design resistances are obtained by dividing the characteristic value of a particular resistance by the global partial safety factor for the resistance, $\gamma_m$. The magnitude of $\gamma_m$ is based on Cl. 6.1 of EN 1993-1-1:2005 and can change depending on the selected National Annex.

Material coefficients for steel in STAAD.Pro take the following default values unless replaced by user’s numerical values provided in the input file.

- Modulus of Elasticity, $E = 205,000$ N/mm$^2$
- Shear Modulus, $G = E/2(1+\nu)$
- Poisson’s Ratio, $\nu = 0.3$
- Unit weight, $\Gamma = 76.8$ KN/m$^3$

The magnitude of design loads is dependent on $\gamma_f$, the partial safety factor for the action under consideration. You are allowed total control in providing applicable values for the factors and their use in various load combinations.
**D5.C.4 Section Classification**

The occurrence of local buckling of the compression elements of a cross-section prevents the development of full section capacity. It is therefore imperative to establish this possibility prior to determining the section capacities. Cross sections are classified in accordance with their geometrical properties and the stress pattern on the compression elements. For each load case considered in the design process, the program determines the section class and calculates the capacities accordingly. It is worth noting that the section class reported in the design output corresponds to the most critical loadcase among those being considered for design.

The EC3 (EN 1993) design module in STAAD can design members with all section profiles that are of Class 1, 2, or 3 as defined in section 5.5 of the code. However, the design of members that have a Class 4 section profile are limited to:

- wide flange
- tee
- single channel
- single angle
- rectangular hollow sections
- circular hollow sections

Also built-up user sections that are class 4 sections are not dealt with in the current version of EC3 design in STAAD.Pro, unless they are defined as any of the section types given above.

The design of laced and battened members is not considered in the current version of EC3 (EN 1993) design module in STAAD.Pro. The current version also does not support the design of tapered section profiles or I-Sections with top and/or bottom plates.

**D5.C.5 Member Design**

EN 1993-1-1:2005, together with any specified National Annex, is used for code check or selection of all cross sections and shapes listed in Section 7C.4 (on page 1757). However, where EN 1993 or the National Annex has not specified a method or values for a specific clause or parameter, STAAD.Pro uses Non-Contradictory Complimentary Information (NCCI) documents as explained in the following corresponding sections.

The design philosophy and procedural logistics are based on the principles of elastic analysis and ultimate limit state design. Two major failure modes are recognized:

- failure by overstressing
- failure by stability considerations

The following sections describe the salient features of the design approach. Members are proportioned to resist the design loads without exceeding the characteristic stresses or capacities. Member selection is done on the basis of selecting the most economic section on the basis of the least weight criteria. It is generally assumed that you (the engineer) will take care of the detailing requirements, such as the provision of stiffeners, and check the local effects like flange buckling, web crippling, etc.

**Note:** The design of class 4 (slender) sections is limited to WIDE FLANGE, TEE, SINGLE CHANNEL, SINGLE ANGLE, and RECTANGULAR & CIRCULAR HOLLOW SECTIONS. The effective section properties are evaluated as described in Cl. 6.2.2.5 of the code.

You are allowed complete control over the design process through the use of the parameters listed in **D5.C.6 Design Parameters** (on page 1776). Default values of parameters will yield reasonable results in most circumstances. However, you should control the design and verify results through the use of the design parameters.
**D5.C.5.1 Members Subject to Axial Loads**

The cross section capacity of tension only members is checked for ultimate limit state as given in Cl. 6.2.3 of the code.

Compression members will be checked for axial capacity of the cross section in addition to lateral buckling/stability. The cross section capacity will be checked as given in section 6.2.4 of the code.

Lateral stability of a pure compression member will be checked as per the method given in Cl. 6.3 of the code. The compression member stability will be verified as:

\[
\frac{N_{Ed}}{N_{b,Rd}} \leq 1.0
\]

Where \(N_{b,Rd}\) is the design buckling resistance given by:

\[
N_{b,Rd} = \frac{x A f_y}{\gamma M1}
\] for Class 1, 2, or 3 cross-sections

\[
N_{b,Rd} = \frac{x A_{eff} f_y}{\gamma M1}
\] for Class 4 cross-sections

Where:

\(\chi\) is the reduction factor as given in section 6.3.12 of the code. The buckling curves used to evaluate the reduction factor are selected from Table 6.2 of the code based on the cross section type and the steel grade.

**Note:** Only the five grades of steel given in Table 6.2 will be used when selecting the buckling curve. The steel grade used for this selection is based on the SGR design input parameter (Refer to D5.C.6 Design Parameters (on page 1776)). Even if you have specified a custom yield strength (using the \(PY\) parameter), the choice of a buckling curve will be based on the value of SGR parameter.

Compression members that are susceptible to torsional or torsional flexural buckling are checked for these modes of failure as well. The non-dimensional slenderness \(\bar{\lambda}_T\) for these members is evaluated per Cl. 6.3.1.4 of the EN 1993 code. The maximum slenderness among the flexural buckling slenderness, torsional slenderness, and torsional-flexural slenderness is used to evaluate the reduction factor, \(\chi\), for such members. The elastic torsional buckling load, \(N_{cr,T}\), and the elastic torsional-flexural buckling load, \(N_{cr,TF}\), are evaluated based on the method given in the NCCI “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” (unless otherwise specified by a particular National Annex). The effective length for the members can be controlled using the \(KZ\), \(KY\), \(LZ\) and \(LY\) parameters. If these parameters are specified, the effective length will be calculated as \(KZ*LY\) for length about the Z-Z axis and \(KY*LY\) for length about the Y-Y axis. By default, the effective length will be taken as the member length.

EN 1993-1-1:2005 does not specifically deal with single angle, double angles, double channels, or Tee sections and does not provide a method to evaluate the slenderness of such members. In these cases, the EC3 (EN 1993) design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 and Table 25 of BS 5950-1:2000 are used in the current version of the Eurocode 3 design module.

**Single Angle Sections**

Angle sections are un-symmetrical and when using BS 5950:2000 table 25 you must consider four axes: two principal, u-u and v-v and two geometric, a-a and b-b. The effective length for the v-v axis, \(L_{vv}\), is taken as the \(L_{VV}\) parameter or \(LY \cdot KY\), if not specified. The a-a and b-b axes are determined by which leg of the angle is fixed by the connection and should be specified using the LEG parameter, see section 5B.6 for more information.
on the LEG parameter. The effective length in the a-a axis is taken as $LY \cdot KY$ and the effective length in the b-b axis as $LZ \cdot KZ$.

The following diagram shows the axes for angles which have been defined with either an ST or RA specification and is connected by its longer leg (i.e., a-a axis is parallel to the longer leg).

| ST angle and USER table angles | RA angle |

**D5.C.5.2 Members Subject to Bending Moments**

The cross section capacity of a member subject to bending is checked as per Cl.6.2.5 of the code. The condition to be satisfied is:

$$\frac{M_{Ed}}{M_{c,Rd}} \leq 1.0$$

Where $M_{c,Rd}$ is the design resistance given by:

- $M_{c,Rd} = M_{pl,Rd} = \frac{W_{pl}f_y}{\gamma_{M0}}$ for class 1 and 2 cross-sections
- $M_{c,Rd} = M_{el,Rd} = \frac{W_{el,\min}f_y}{\gamma_{M0}}$ for class 3 cross-sections
- $M_{c,Rd} = \frac{W_{eff,\min}f_y}{\gamma_{M0}}$ for class 4 cross-sections

Cross sectional bending capacity checks will be done for both major and minor axis bending moments.

Members subject to major axis bending will also be checked for Lateral Torsional Buckling resistance as per Section 6.3.2 of the code. The design buckling resistance moment $M_{b,Rd}$ will be calculated as:

$$M_{b,Rd} = XLTW_yf_y\gamma_{M1}$$
Where:

$\chi_{LT}$ is the reduction factor for lateral torsional buckling. This reduction factor is evaluated per Cl. 6.3.2.2 or Cl 6.3.2.3 of the EN 1993 code depending on the section type. For I sections, the program will by default use Cl. 6.3.2.3 to evaluate $\chi_{LT}$ and for all other sections the program will resort to Cl 6.3.2.2. However, if a particular National Annex has been specified, the program will check if the National Annex expands on Cl6.3.2.3 (Table 6.5) to include sections other than I sections. If so, the program will use Cl. 6.3.2.3 for the cross-section(s) included in Cl 6.2.2.3 (or Table 6.5). For all other cases the program will use Cl. 6.3.2.2.

**Note:** You have the option to choose the clause to be used to calculate $\chi_{LT}$ through the MTH design parameter. Setting MTH to 0 (default value) will cause the program to choose Cl.6.3.2.3 for I Sections and Cl 6.2.3.2 for all other section types. As mentioned above, if the National Annex expands on Cl. 6.3.2.3 to include sections other than I Sections, the program will use Cl. 6.3.2.3 by default.

When using Cl. 6.3.2.3 to calculate $\chi_{LT}$, the program will consider the correction factor $kc$ (Table 6.6 of EN 1993-1-1:2006) based on the value of the KC parameter in the design input. By default the value of KC will be taken as 1.0. If you want the program to calculate kc, you must explicitly set the value of the KC parameter to zero.

**Note:** If the National Annex specifies a different method to calculate kc (e.g. the British, Singapore & Polish NAs), the program will use that method by default even if the KC parameter has not been explicitly set to zero. If the NA method does not deal with a specific condition while working out kc, the program will then fall back to table 6.6 of the code, thus ensuring that kc is considered for the particular NA.

The non-dimensional slenderness $\bar{\lambda}_{LT}$ (used to evaluate $\chi_{LT}$) for both the above cases is evaluated as:

$$\bar{\lambda}_{LT} = \sqrt{\frac{Wy^2}{M_{cr}}}$$

Where:

$M_{cr}$ is the elastic critical moment for lateral torsional buckling. EN 1993-1-1 does not however specify a method to evaluate $M_{cr}$. Hence, the program will make use of the method specified in Annex F of DD ENV 1993-1-1 to evaluate $M_{cr}$ by default.

**Note:** The method specified in Annex F will be used only when the raw EN 1993-1-1:2005 code is used without any National Annex. If a National Annex has been specified, the calculation of $M_{cr}$ (and $\bar{\lambda}_{LT}$) will be done based on the specific National Annex. (Refer to **D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005]** (on page 1792) for specific details). If the National Annex does not specify a particular method or specify a reference document, the program will use the NCCI document SN-003a-EN-EU for doubly symmetric sections and SN030a-EN-EU for mono-symmetric sections that are symmetric about their weak axis. For all other sections types the program will use Annex F of DD ENV 1993-1-1 to calculate $M_{cr}$. In cases where Annex F does not provide an adequate method to evaluate $M_{cr}$, such as for Channel sections, the program will resort to the method as per Cl4.3.6 of BS 5950-1:2000 to calculate the lateral torsional buckling resistance moment ($Mb,Rd$) for the member.

**D5.C.5.3 Members Subject to Shear**

The cross section capacity of a member subject to shear is checked as per Cl. 6.2.6 of the code. The condition to be satisfied is:

$$\frac{V_{Ed}}{V_{c,Rd}} \leq 1.0$$
where
\[ V_{c,Rd} = \text{the is the shear design resistance given by:} \]
\[ V_{c,Rd} = V_{pl,Rd} = \frac{A_v(f_y/\sqrt{3})}{\gamma M_0} \]
\[ A_v = \text{the shear area and is worked out for the various section types as given in Cl. 6.2.6(3) of the code} \]

Shear Buckling

For sections that are susceptible to shear buckling, the program will perform the shear buckling checks as given in Section 5 of EN 1993-1-5. The shear buckling checks will be done only for I-Sections and Channel sections. Shear stresses induced from torsional loads are taken into account while performing torsion checks.

**Note:** Web shear buckling is checked in STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) and later.

The susceptibility of a section to shear buckling will be based on the criteria given in Cl. 5.1(2) of EN 1993-1-5 as is as given as follows:

**a.** For unstiffened webs, if \( h_w / t > 72 \times \varepsilon / \eta \), the section must be checked for shear buckling.

The design resistance is calculated as:
\[ V_{b,Rd} = V_{bw,Rd} \leq \frac{\eta f_y h_w t}{\sqrt{3} \gamma M_1} \]
\[ V_{bw,Rd} = \chi w f_y h_w t \]

where
- \( h_w = \text{distance between flanges of an I Section (i.e., depth - 2x flange thickness)} \)
- \( t = \text{thickness of the web} \)
- \( \varepsilon = \sqrt{(235/f_y)}, \text{where } f_y \text{ is the yield stress} \)
- \( \eta = 1.2 \text{ for steel grades up to and including S 460 and} \)
  \( = 1.0 \text{ for other steel grades} \)
- \( k_t = \text{as defined in sections below} \)
- \( \chi w = \text{the web contribution factor obtained from Table 5.1 of the EC3 code and is evaluated per the following table:} \)

**Table 144: Evaluate of \( \chi w \)**

<table>
<thead>
<tr>
<th>Slenderness Parameter</th>
<th>Rigid End Post</th>
<th>Non-rigid End Post</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \chi w &lt; 0.83 / \eta )</td>
<td>( \eta )</td>
<td>( \eta )</td>
</tr>
<tr>
<td>( 0.83 / \eta \leq \chi w &lt; 1.08 )</td>
<td>( 0.83 / \chi w )</td>
<td>( 0.83 / \chi w )</td>
</tr>
<tr>
<td>( \chi w \geq 1.08 )</td>
<td>( 1.37 / 0.7 + \chi w )</td>
<td>( 0.83 / \chi w )</td>
</tr>
</tbody>
</table>

\[ \chi w = \frac{h_w}{86.4 \times t \times \varepsilon} \]

**b.** For stiffened webs, if \( h_w / t > 31 \times E k_t / \eta \), the section must be checked for shear buckling.
The design resistances considers tension field action of the web and flanges acting as struts in a truss model. This is calculated as:

\[ V_{b, Rd} = V_{bw, Rd} + V_{bf, Rd} \leq \frac{\eta f_{yw} h_w t}{\sqrt{3} M_1} \]

Where:

\[ V_{bf, Rd} = \frac{h_f t_f^2 f_{yf}}{\sigma_f M_1} \left[ I - \left( \frac{M_{Ed}}{M_{f, Rd}} \right)^2 \right] \]

\[ b_f = \text{the width of the flange which provides the least axial resistance, not to be taken greater than } 15e_t \text{ on each side of the web} \]

\[ t_f = \text{the thickness of the flange which provides the least axial resistance} \]

\[ M_{f, Rd} = \frac{M_{f,k}}{\gamma M_0}, \text{ the moment of resistance of the cross section consisting of the effective area of the flanges only. For a typical I Section or PFD, this is evaluated as } b \ t_f \ h_w. \]

When an axial load, \( N_{Ed} \), is present, the value of \( M_{f, Rd} \) is reduced by multiplying by the following factor:

\[ 1 - \frac{N_{Ed}}{\left( A_{f1} + A_{f2} \right) f_{yf}} \]

\[ A_{f1}, A_{f2} = \text{the areas of the top and bottom flanges, respectively} \]

\[ c = a \left( 0.25 + \frac{1.6 b_f t_f^2 f_{yf}}{h_f t_f h_w} \right) \]

\[ a = \text{transverse stiffener spacing. The equation of } c \text{ is likewise used to solve for a sufficient stiffener spacing in the case of demand from loads exceeding the calculated capacity for a specified stiffener spacing} \]

The following equation must be satisfied for the web shear buckling check to pass:

\[ \eta_3 = \frac{V_{Ed}}{V_{b, Rd}} \leq 1.0 \]

where

\[ V_{Ed} = \text{the design shear force} \]

**Note:** The shear forces due to any applied torsion will not be accounted for if the TOR parameter has been specifically set to a value of 0 (i.e., ignore torsion option).

If the stiffener spacing has not been provided (using the STIFF parameter), then the program assumes that the member end forms a non-rigid post (case c) and proceeds to evaluate the minimum stiffener spacing required.

**D5.C.5.4 Members Subject to Torsion**

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 2) build 2007.07 or later.
Eurocode 3 (EN 1993-1-1:2005) gives very limited guidance for the analysis and design of torsion members. While both elastic and plastic analyses are permitted generally, the design analysis methods for torsion discussed within EC3 are primarily based on elastic methods. Also, only the first yield design resistance is specifically discussed for torsion members. Furthermore, there is no guidance on section classification nor on how to allow for the effects of local buckling on the design resistance for combined torsional effects. EC3 also does not specifically deal with members subject to combined bending and torsion and loosely states that the yield criteria (Eqn 6.1 in the code) can be used for elastic verification.

The method used by STAAD.Pro is therefore based on the SCI publication "P057: Design of members subject to combined bending and torsion". Though this publication is based on the British standard BS 5950-1, the principles from this document are applied in the context of Eurocode 3.

**Note:** At the time this feature has been implemented in STAAD.Pro, SCI are in the process of updating document P057 to be in accordance with Eurocode 3. Hence this method might be subject to modifications subject to the publication of a newer version of P057. The NCCI document "SN007b-EN-EU: Torsion" will also be referenced where appropriate.

**Code Basis**

Torsion design in EC3 is given in Cl. 6.2.7 of EN 1993-1-1:2005. Therefore, this clause is used primarily for this implementation.

EN 1993-1-1:2005 does not deal with members subject to the combined effects of torsion and lateral torsional buckling. However, EN 1993-1-6 considers such a condition in Appendix A. Therefore, STAAD.pro uses Appendix A of EN 1993-1-6 to check for members subject to combined torsion and LTB.

The following clauses from EC3 are then considered:

- Cl. 6.2.7(1)
- Cl. 6.2.7(9)
- Cl. 6.2.7(5)
- EC-3-6 App A

**Note:** STAAD.Pro does, however, use this clause (6.2.7) to report the output for all torsion checks. Also any distortional deformations and any amplification in the torsional or shear stresses due to distortions will be neglected by the program.

- Clause 6.2.7(1)

  States that for members subject to torsion, the design torsional moment $T_{Ed}$ at each cross section should satisfy:

  $$
  T_{Ed} / R_{Rd} \leq 1.0
  $$

  Where:

  $T_{Rd}$ is the design torsional resistance of the cross section.

  This is the primary condition that will need to be satisfied for members subject to torsion. The method for working out the torsional resistance $T_{Rd}$ for the various cases is dealt in the following sections.

- Cl. 6.2.7(9)

  States that:
For combined shear force and torsional moment, the plastic shear resistance accounting for torsional effects should be reduced from $V_{pl,Rd}$ to $V_{pl,T,Rd}$ and the design shear force should satisfy:

$$V_{Ed} / V_{pl,T,Rd} \leq 1.0$$

The code also gives means to evaluate $V_{pl,T,Rd}$ in equations 6.26 to 6.28. These equations, however, only deal with I/H sections, Channel sections, and structural hollow sections (RHS, SHS, CHS). Therefore, the application of Cl. 6.2.7(9) is only performed for these section profiles.

- **Cl 6.2.7(5)**

  States that the yield criteria given in Cl. 6.2.1(5) of EN 1993-1-1:2005 may be used for elastic verification. STAAD.Pro evaluates the stresses due to the various actions on the cross section and applies this yield criterion.

The program allows for two types of checks for members subject to torsion for EC3 design:

**I. Basic Stress Check**

This method is intended to be a simplified stress check for torsional effects. This method will produce the output corresponding to Cl. 6.2.7(5) of EN 1993-1-1.

**II. Detailed Checks**

This method will perform a full torsional analysis of the member. All four of the clause checks mentioned earlier will be performed.

The details of these checks are as described below.

You have the option to choose the method to be used for a specific member or group of members. This will be facilitated by setting the value of the TORSION. The TORSION parameter set to zero by default, which results in torsion checks only being performed if the member is subject to torsional moments (i.e., for this default setting, the program will ignore torsion checks if there is no torsional moment in the member). Setting the value of the TORSION parameter to three (3) will cause the program to ignore all torsional moments. The detailed output (i.e., TRACK 2) will indicate that torsion has been ignored for that particular member. The details of setting the values to one (1) or two (2) and the corresponding checks performed are as described below. Refer to D5.C.6 Design Parameters (on page 1776) for additional details.

**Note:** If the TORSION parameter is set to 1 or 2, the program will perform the appropriate checks even if the member is not subject to torsional moments. In such cases, the program will perform the checks with a value of zero for the torsional moment.

**D5.C.5.4.1 Basic stress check**

This method is used when the TORSION parameter is specified as one (1).

This method is intended to be a simplified stress check for torsional effects per Cl. 6.2.7(5). Any warping stresses that may develop due to the end conditions will be ignored for this option. The program will consider the forces (including torsion) at various sections along the length of the member and for each section, will calculate the resultant stress (Von Mises) at various points on the cross section. The location and number of points checked for a cross section will depend on the cross section type and will be as described below.

The stress check will be performed using equation 6.1 of EN 1993-1-1:2005 as given below:

$$\left(\frac{\sigma_{x,Ed}}{f_{y}/\gamma_{M0}}\right)^2 + \left(\frac{\sigma_{z,Ed}}{f_{y}/\gamma_{M0}}\right)^2 - \left(\frac{\sigma_{x,Ed}}{f_{y}/\gamma_{M0}}\right)\left(\frac{\sigma_{z,Ed}}{f_{y}/\gamma_{M0}}\right) + 3\left(\frac{\tau_{Ed}}{f_{y}/\gamma_{M0}}\right)^2 \leq 1$$

where

- $\sigma_{x,Ed}$ = the longitudinal stress
- $\sigma_{z,Ed}$ = the transverse stress and
- $\tau_{Ed}$ = the resultant shear stress.
**Note:** Since transverse stresses are very small under normal loading conditions (excluding hydrostatic forces), the term will be negligible and hence is taken as zero.

\[
\sigma_{x,Ed} = \sigma_x + \sigma_{bz} + \sigma_{by} = \frac{F_x}{A_x} + \frac{M_z}{Z_z} + \frac{M_y}{Z_y}
\]

\[
\tau_{Ed} = \frac{T}{J} \cdot t + \frac{V_y}{I_z} \cdot Q/(I_z^2 t) + \frac{V_z}{I_y} \cdot Q/(I_y^2 t)
\]

where

- \( T \) = the torsion at the particular section along the length of the member
- \( J \) = the torsion constant
- \( t \) = the thickness of the web/flange
- \( V \) = the shear force
- \( Q \) = the statrical moment about the relevant axis
- \( I \) = the second moment of area about the relevant axis

The stress check as per equation 6.1 is performed at various stress points of a cross section as shown in figures below:

<table>
<thead>
<tr>
<th>Shape</th>
<th>Section Sketch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Doubly symmetric wide flange profile</td>
<td><img src="image1" alt="Doubly Symmetric Wide Flange Profile" /></td>
</tr>
<tr>
<td>Pipe profiles</td>
<td><img src="image2" alt="Pipe Profiles" /></td>
</tr>
</tbody>
</table>

\[ \alpha = \tan^{-1}(M^x/M_y) \]
The resultant ratio will be reported under Cl. 6.2.7(5) in the detailed design output.

D5.C.5.4.2 Detailed stress check
This method is used when the TORSION parameter is specified as two (2).

This method performs a detailed torsional analysis of a member depending on the torsion loading conditions and the support conditions at the member ends. This method is based on the SCI publication P057 and includes any warping stresses (direct warping stresses and warping shear stresses) depending on the end conditions of the member. This implementation considers seven different cases of loading and end conditions as given in...
The loading/end conditions for a member are specified by the use of the CMT design parameter (Refer to D5.C.6 Design Parameters (on page 1776) for parameter values and descriptions).

All the equations used to evaluate the torsional moments and associated stresses are as given in Appendix B of P057. The resultant stresses are evaluated at various sections along the length of the member and the following checks will be performed:

Clause 6.2.7(1) – Torsional resistance of the section.

In general, the torsion at any section $T_{Ed}$ is resolved into two components, viz.

The pure torsional (St. Venant’s) moment ($T_{t,Ed}$) and

The warping torsional moment ($T_{w,Ed}$)

Therefore,

$$T_{Ed} = T_{t,Ed} + T_{w,Ed} = GJ\phi' = EH\phi''$$

where

$\phi'$ and $\phi''$ are the first and third derivatives of twist ($\phi$), respectively, and depend on the end conditions and loading. These are evaluated from the equations in Annex B of P057 and are based the specified CMT parameter.

Note: Although the equation given the NCCI document SN007b-EN-EU can be used to evaluate $T_{wrd}$, the NCCI does not give the eqn. to evaluate $\phi'''$. Therefore, Annex B of P057 is used.

The torsional resistance of the section is also considered as the sum of the pure torsion resistance and the warping torsion resistance. The pure torsion resistance ($T_{t,Rd}$) and the warping torsion resistance ($T_{w,Rd}$) are evaluated as:

For closed sections:

$$T_{t,Rd} = 2 \cdot A_c \cdot t \cdot \tau_{max}$$

where

$A_c$ = the area enclosed by the mean perimeter
$t$ = the max thickness
$\tau_{max}$ = the max. allowable shear stress $= (f_y/\sqrt{3})/\Gamma_m0$

For open sections (I & channel):

$$T_{t,Rd} = \tau_{max} \cdot J / t$$

where

$J$ = the torsion const
$t$ = the max thickness

$T_{w,Rd} = (f_y/\Gamma_{m0}) \cdot t \cdot b^2 / 6$

where

$b$ = the width of the section
$t$ = the thickness of the flange for I- sections; minimum of flange or web thickness channel sections

The check according to Cl 6.2.7(1) will then be performed to ensure that the following conditions are satisfied:

$$T_{t,Ed} / T_{t,Rd} \leq 1$$
$$T_{w,Ed} / T_{w,Rd} \leq 1$$
$$T_{Ed} / T_{Rd} \leq 1$$
Clause 6.2.7(9) – Plastic shear resistance due to torsion

STAAD.Pro checks for shear resistance of a section based on Cl. 6.2.6 for EC3 and the plastic shear resistance (in the absence of torsion) is evaluated as:

\[ V_{pl,Rd} = \frac{A_v (f_y / \sqrt{3})}{\gamma M_0} \]

where

\[ A_v = \text{as pre Cl.6.2.6 (3) for the various sections} \]

When torsion is present, along with the shear force, the design shear resistance will be reduced to \( V_{pl,T,Rd} \), where \( V_{pl,T,Rd} \) is evaluated as follows:

i. For I or H Sections:

\[ V_{pl,T,Rd} = \sqrt{1 - \frac{\tau_{t,Ed}}{1.25 (f_y / \sqrt{3}) / \gamma M_0} V_{pl,Rd}} \]

ii. For Channel Sections:

\[ V_{pl,T,Rd} = \left[ 1 - \frac{\tau_{t,Ed}}{1.25 (f_y / \sqrt{3}) / \gamma M_0} - \frac{\tau_{w,Ed}}{(f_y / \sqrt{3}) / \gamma M_0} \right] V_{pl,Rd} \]

iii. For Structural Hollow Sections:

\[ V_{pl,T,Rd} = \left[ 1 - \frac{\tau_{t,Ed}}{(f_y / \sqrt{3}) / \gamma M_0} \right] V_{pl,Rd} \]

where

\[ \tau_{t,Ed} = \text{the shear stress due to direct (St. Venant’s) torsion} \]
\[ \tau_{w,Ed} = \text{the shear stress due to warping torsion} \]

The various shear stresses due to torsion \( \tau_{t,Ed} \) and \( \tau_{w,Ed} \) are evaluated as follows:

i. For Closed sections:

The shear stresses due to warping can be ignored as they will be insignificant and hence:

\[ \tau_{t,Ed} = \frac{T_{Ed}}{2 A_c t} \] [Ref NCCI Sn007b-EN-EU]

where

\[ T_{Ed} = \text{the applied torsion} \]
\[ A_c = \text{the area delimited by the mean perimeter} \]
\[ t = \text{the thickness of the cross section} \]
\[ \tau_{w,Ed} = 0, \text{since warping is ignored} \]

ii. For Open sections [I, H, Channel] sections:

For I and H sections, the web will not be subject to warping stresses and therefore warping shear can be ignored (\( \tau_{w,Ed}=0 \)).

The stress due to pure torsion is evaluated as:

\[ \tau_{t,Ed} = G t \varphi' \] [Ref SCI pub. P057]

where

\[ G = \text{the shear modulus} \]
\[ \varphi' = \text{a function depending on the end condition and loading(T). This will be taken from section 6 and Annex B of P057.} \]
Note: Although the maximum stress is at the thickest section of the profile, the program uses the web thickness for this clause (since the shear capacity is based on the web area) unless the load is parallel to the flanges, in which case the flange thickness is used.

For channel sections that are free to warp at the supports and, thus, are not subject to warping stresses:

The warping shear stress is evaluated as:

$$\tau_{w,Ed} = E \cdot S_w \cdot \phi''' / t$$  \hspace{1cm} \text{[Ref SCI pub. P057]}$$

where

- $E$ = the elastic modulus
- $S_w$ = the warping statistical moment
- $\phi'$ = a function depending on the end condition and loading(T). This will be taken from section 6 and Annex B of P057.

Clause 6.2.7(5) – Check for elastic verification of yield

Eurocode 3 gives yield criterion as per eqn. 6.1 and STAAD.Pro uses the yield criterion given in EC-3. When a member is subject to combined bending and torsion, some degree of interaction occurs between the two effects. The angle of twist caused by torsion is amplified by the bending moments and will induce additional warping moments and torsional shears. Account must also be taken of the additional minor axis moments produced by the major axis moments acting through the torsional deformations, including the amplifications mentioned earlier.

For members subject to bending and torsion, the stresses are evaluated as follows:

Direct bending stress (major axis):
$$\sigma_{bz} = \frac{M_z}{Z_z}$$

Direct bending stress (minor axis):
$$\sigma_{by} = \frac{M_y}{Z_y}$$

Direct stress due to warping:
$$\sigma_w = E \cdot W_{ns} \cdot \phi''$$

Direct stress due to twist (min. axis):
$$\sigma_{byt} = \frac{M_{yt}}{Z_y}$$

Direct stress due to axial load (if any):
$$\sigma_c = \frac{P}{A}$$

where

- $M_z$ = the major axis moment & $M_y$ is the minor axis moment
- $\phi''$ = the differential function based on twist (ref P057 Annex B. & Table 6)
- $W_{ns}$ = the normalized warping function
- $M_{yt}$ = $\varphi \cdot M_z$ (see Appendix B of P057 to evaluate $\varphi$)

Shear stresses due to torsion and/or warping is evaluated as described above for Clause 6.2.7(9).

Check for yield (capacity checks) is then done according to Eqn 6.1 of EN 1993-1-1:2005, as described for the Basic Stress Check (TORSION = 1):

$$\left(\frac{\sigma_{x,Ed}}{f_y/\gamma_{M0}}\right)^2 + \left(\frac{\sigma_{z,Ed}}{f_y/\gamma_{M0}}\right)^2 - \left(\frac{\sigma_{x,Ed}}{f_y/\gamma_{M0}}\left(\frac{\sigma_{z,Ed}}{f_y/\gamma_{M0}}\right) + 3\left(\frac{\tau_{Ed}}{f_y/\gamma_{M0}}\right)^2 \leq 1$$

Clause EC-3:6 App A – Check for combined Torsion and Lateral Torsional buckling

The interaction check due to the combined effects of bending (including lateral torsional buckling) and torsion will be checked using Annex A of EN 1993-6: 2007. Note that this interaction equation does not include the effects of any axial load.

Caution: At present, SCI advises that no significant work has been published for this case and work is still ongoing. So at present is advisable not to allow for torsion in a member with large axial load.

Members subject to combined bending and torsion will be checked to satisfy:
\[ \frac{M_{y,ED}}{\chi_{LT}M_{y,Rk}/\gamma M_1} + \frac{C_{Mz}M_{z,Ed}}{M_{z,Rk}/\gamma M_1} + \frac{k_w k_{zw} k_a T_{w,Ed}}{T_{w,Rk}/\gamma M_1} \leq 1 \]

where

- \( C_{mz} \) = the equivalent uniform moment factor for bending about the z-z axis, according to EN 1993-1-1 Table B.3.
- \( k_w = 0.7 - \frac{0.2 T_{w,Ed}}{T_{w,Rk}/\gamma M_1} \)
- \( k_{zw} = 1 - \frac{M_{z,Ed}}{M_{z,Rk}/\gamma M_1} \)
- \( k_a = \frac{1 - M_{y,Ed}/M_{y,cr}}{1} \)
- \( M_{y,Ed} \) and \( M_{z,Ed} \) = the design values of the maximum moment about the y-y and z-z axis, respectively.
- \( M_{y,Rk} \) and \( M_{z,Rk} \) = are the characteristic values of the resistance moment of the cross-section about y-y and z-z axis, respectively, from EN 1993-1-1, Table 6.7.
- \( M_{y,cr} \) = the elastic critical lateral-torsional buckling moment about the y-y axis.
- \( T_{w,Ed} \) = the design value of the warping torsional moment.
- \( T_{w,Rk} \) = the characteristic value of the warping torsional resistance moment.
- \( \chi_{LT} \) = the reduction factor for lateral torsional buckling according to 6.3.2 of EN 1993-1-1.

**Note:** For all of the above checks the effective length of the member to be used for torsion can be set by using the EFT design parameter.

### D5.C.5.5 Members Subject to Combined Forces

**Members subject to Bending and Axial Force**

When a member is subject to a combined axial load and a bending moment, the program evaluates a reduced moment capacity based on Cl. 6.2.9 of the code. For Class 1, 2, and 3 sections, the program evaluates the reduced moment from the equations given in Cl. 6.2.9.1 of the code. For class 4 sections, the interaction equation given by equation 6.44 are checked.

In the case of members subject to axial load and biaxial bending, the program will consider the interaction equation 6.41 of the code.

**Note:** By default, the program will use the values of the constants “\( \alpha \)” and ‘\( \beta \)’ as given in the code for the different sections types. However, you can override these values using the ALPHA and BETA design parameters (Refer to D5.C.6 Design Parameters (on page 1776)).

**Note:** The program uses the parameter ELB (Refer to D5.C.6 Design Parameters (on page 1776)) to override the Cl.6.2.9 checks for combined axial load and bending case. When specified as 1, the program uses the more general equation 6.2 of EN 1993-1-1, instead.

**Members subject to Bending, Shear, and Axial Force**

When a member is subject to a combined axial load, shear force, and a bending moment, the program evaluates the reduced yield strength as given in Cl 6.2.10 (3) of the code. The reduction in the yield strength is done only
when the applied shear force exceeds 50% of the design shear resistance $V_{pl,Rd}$. This reduced yield strength is then used to evaluate the reduced moment capacity of the section.

Members subject to Bending and Axial Compression

The bending resistance of members could be reduced by the presence of a co-existent axial load. This is then checked against the lateral-torsional buckling resistance of the section. The EN 1993 design module in STAAD takes such a scenario into account and performs the necessary checks as per Cl. 6.3.3 of the code.

Generally, EC3 requires checking cross-section resistance for local capacity and also checking the overall buckling capacity of the member. In the case of members subject to axial tension and bending, there is provision to take the stabilizing effect of the tension load into consideration. This is achieved by modifying the extreme compression fibre stress and calculating an effective applied moment for the section. The EN 1993-1-1 code in STAAD.Pro checks to ensure that both the interaction equations 6.61 and 6.62 of the code are satisfied. The interaction factors $k_{zz}$, $k_{yy}$, $k_{zy}$ & $k_{yz}$ will be evaluated using Annex B of EN 1993-1-1 by default. Hence for the EN 1993-1-1 code in STAAD.Pro (without National Annexes), uses Annex B. The choice between using Annex A and Annex B will be based on the choice specified by a particular National Annex, if used. If the National Annex itself gives a choice between Annex A and Annex B, the program uses Annex B to evaluate the interaction factors.

**Note:** EN 1993-1-1:2005 does not specifically deal with single angle, double angles, double channels or Tee sections and does give a method to evaluate the slenderness of such members. In these cases, the Eurocode 3 (EN 1993-1-1) design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 of BS 5950-1:2000 is used in the current version of the EC3 design module. See "Single Angel Sections" (on page 1758) for ST and RA angle specifications.

**Note:** Laced or battened compression members are *not* dealt within the current version of EC3 (EN 1993) design module in STAAD.Pro.

**D5.C.5.6 Design of Slender pipe sections to EN 1993-1-6**

The design of Slender CHS sections is performed per EN 1993-1-6:2007 (hereafter, EC3-6). EC3-6 does not specify additional or modified safety factors. Therefore, the program uses the default safety factors from EN 1993-1-1.

**Note:** You can change these values through the GM0, GM1, & GM2 design parameters.

EC3-6 deals with four types of ultimate limits states: plastic limit state, cyclic capacity limit state, buckling limit state, and fatigue. The following are considered by STAAD.Pro:

- **LS1 - Plastic limit state:** Deals with the condition when the capacity of the structure is exhausted by yielding of the material.
- **LS3 - Buckling Limit state:** Deals with the condition in which the structure (or shell) develops large displacements normal to the shell surface, caused by loss of stability under compressive and/or shear membrane stresses.

The limit state verification is made based on the "Stress design" method described in EC3-6. The stress design approach takes into account three categories of stresses:

- **Primary stresses:** Stresses that are generated for the member to be in equilibrium with the direct imposed loads.
- **Secondary stresses:** Those that are generated for internal compatibility or for compatibility at supports due to imposed loads or displacements (e.g., temperature, settlement etc.)
• Local stresses: Local stresses generated due to cyclic loading (or fatigue).

Only the primary stresses are considered the program. The primary stresses considered are those generated due to axial loads, bending, shear and/or a combination of these conditions.

**Note:** In the context of slender pipe section design for the Eurocode 3 module, the secondary and local stresses can be neglected since the loads and corresponding stresses dealt with in the design engine are largely direct and shear stresses.

The local axis coordinate system for a CHS is defined as:

<table>
<thead>
<tr>
<th>Axis</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>circumferential</td>
<td>around the circumference of the circular cross section (θ)</td>
</tr>
<tr>
<td>meridional</td>
<td>along the length of the member (x)</td>
</tr>
<tr>
<td>normal</td>
<td>perpendicular to the tangential plane formed by the circumferential and meridional directions (n)</td>
</tr>
</tbody>
</table>

and the corresponding membrane stresses will follow the convention given below:

![Figure 178: Nomenclature for membrane and transverse stresses in Slender CHS sections](image)

**Stress Design**

Stress checks are made based on the “Stress design” method as per Section 8.5 of the code. This section deals with the buckling strength of the member (LS3). The principle is to evaluate the membrane stresses due to the applied loads and then compare that to the buckling strength, which is evaluated giving due consideration for local buckling effects.

The membrane stresses are evaluated as given in Annex A of the code. The pipe section is considered as an unstiffened cylindrical shell.

i. Meridional Stresses:

a. Axial load
\[ F_x = 2 \pi r P_x \]
\[ \sigma_x = F_x / (2 \pi r t) \]

**b. Axial stress from bending**

\[ M = \pi r^2 P_{x,\text{max}} \]
\[ \sigma_x = \pm M / (\pi r^2 t) \]

**ii. Shear Stress:**

**a. Transverse force, \( V \)**

\[ V = \pi r P_{\theta,\text{max}} \]
\[ \tau_{\text{max}} = \pm V / (\pi r t) \]

**b. Shear from torsional moment, \( M \)**

\[ M_t = 2 \pi r^2 P_\theta \]
\[ \tau = M_t / (2 \pi r^2 t) \]

Where:

\( r \) is the radius of the middle surface of the shell wall.
\( t \) is the wall thickness of the cylinder

Calculation of Axial Buckling Stress

The buckling strength of a slender pipe section is evaluated using the method given in section 8.5.2 of EC3-6. The design buckling stresses (buckling resistance) are calculated separately for axial, circumferential, and shear. The circumferential stresses are ignored in STAAD.Pro.

The naming convention and the coordinate axis used will be as given in the following diagram:

*Figure 179: Naming convention and coordinate system used for the buckling stress of a slender CSH section*

The axial buckling resistance is given by:

\[ \sigma_{x,Rd} = \sigma_{x,Rk} / \Gamma_{M1} \]

**Note:** \( \Gamma_{M1} \) will have the same default value of 1.0 as in EN 1993-1-1.

\( \sigma_{x,Rk} \) is the characteristic buckling strength given by:
\( \sigma_{x,Rk} = X_x \cdot f_{yk} \)

Where:

\( \chi_x \) is the meridional buckling reduction factor. \( \chi_x \) is evaluated per Section 8.5.2(4) of EC3-6 and is determined as a function of the relative shell slenderness given by:

\[
\lambda_x = \sqrt{\frac{f_{yk}}{\sigma_{x,cr}}} \]

Where:

\( \sigma_{x,cr} \) is the elastic buckling critical stress.

Once the relative slenderness is evaluated, the reduction factor is calculated as follows:

\[
\chi = 1 \text{ when } \lambda \leq \lambda_0 \]

\[
\chi = 1 - \beta \left( \frac{\lambda - \lambda_0}{\lambda_P - \lambda_0} \right)^{\eta} \text{ when } \lambda_0 < \lambda < \lambda_P \]

\[
\chi = \frac{a}{\lambda^2} \text{ when } \lambda_P \leq \lambda
\]

where

\[
\lambda_P = \sqrt{\frac{a}{1-\beta}}
\]

The meridional buckling parameters the factors \( \alpha \) and \( \beta \) are evaluated per section D.1.2.2 of EC3-6.

**Note:** A "Normal" fabrication quality will be assumed when evaluating the fabrication quality parameter as given in table D.2 of the code, unless the fabrication quality is set using the FAB design parameter. Refer to D5.C.6 Design Parameters (on page 1776)

The elastic critical buckling stress, \( \sigma_{x,cr} \) and the factors \( \alpha \) and \( \beta \) are evaluated per Annex D of EC3-6. The details are as given below:

The CHS section is classified based on the following criteria:

<table>
<thead>
<tr>
<th>CHS Length Classification</th>
<th>Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Short</td>
<td>( \omega \leq 1.7 )</td>
</tr>
<tr>
<td>Medium</td>
<td>( 1.7 &lt; \omega \leq 0.5 \cdot r/t )</td>
</tr>
<tr>
<td>Long</td>
<td>( \omega &gt; 0.5 \cdot r/t )</td>
</tr>
</tbody>
</table>

Where:

\[
\omega = \frac{l}{\sqrt{rt}}
\]

The elastic critical buckling critical stress is evaluated as:

\[
\sigma_{x,Rcr} = 0.605 \cdot E \cdot C_x \cdot (t/r)
\]

Where:
Calculation of Shear Buckling Stress

The shear buckling resistance is given by:

\[ \tau_{x\theta, Rd} = \frac{\tau_{x\theta, Rk}}{\gamma_M} \]

**Note:** \( \gamma_M \) will have the same default value of 1.0 as in EN 1993-1-1.

\( \tau_{x\theta, Rk} \) is the characteristic buckling shear strength given by:

\[ \tau_{x\theta, Rk} = \chi_{\theta} \cdot f_{yk} \]

Where:

\( \chi_{\theta} \) is the shear buckling reduction factor. \( \chi_{\theta} \) will be worked out as given in section 8.5.2(4) of En 1993-1-6 and is determined as a function of the relative shell slenderness given by:

\[ \lambda_{\theta} = \sqrt{\frac{f_{yk}}{\tau_{x\theta, cr}}} \]

Where:

\( \tau_{x\theta, cr} \) is the elastic buckling critical stress.

The reduction factor, \( \chi_{\theta} \), is then evaluated as described for the axial buckling stress, based on the same \( \chi_p \), \( \alpha \), and \( \beta \) parameters given in Annex D of EC3-6.

The CHS section is classified based on the following criteria:

<table>
<thead>
<tr>
<th>CHS Length Classification</th>
<th>Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Short</td>
<td>( \omega \leq 10 )</td>
</tr>
<tr>
<td>Medium</td>
<td>( 10 &lt; \omega \leq \frac{8.7 \cdot r}{t} )</td>
</tr>
<tr>
<td>Long</td>
<td>( \omega &gt; \frac{8.7 \cdot r}{t} )</td>
</tr>
</tbody>
</table>

Where:

\[ \omega = \frac{l}{\sqrt{\pi}} \]

The elastic critical buckling critical stress is evaluated as:

\[ \tau_{x\theta, cr} = 0.75EC_r \sqrt{\frac{4}{\omega \cdot r}} \]

Where:

\( C_r \) is a factor dependant upon whether the CHS length classification as described in section D.1.4.1 of EC-3-6. A "Normal" fabrication quality will be assumed when working out the fabrication quality parameter as given in table D.6 of the code, unless the fabrication quality is set using the FAB design parameter.
Buckling Strength Verification

The buckling strength verification will be performed so as to satisfy the following conditions:

For axial stresses:

\[ \sigma_{x,Ed} \leq \sigma_{x,Rd} \]

For shear stresses:

\[ \tau_{x\theta,Ed} \leq \tau_{x\theta,Rd} \]

For a combined case of axial and shear stresses acting together, an interaction check will be done according to equation 8.19 of the code as below:

\[
\left( \frac{\sigma_{x,Ed}}{\sigma_{x,Rd}} \right)^{k_x} + \left( \frac{\tau_{x\theta,Ed}}{\tau_{x\theta,Rd}} \right)^{k_\tau} \leq 1
\]

Where:

\[ k_x \text{ and } k_\tau \text{ are the interaction factors as given in section D.1.6 of EN 1993-1-6:} \]

\[ k_x = 1.25 + 0.75 \cdot X_x \]
\[ k_\tau = 1.75 + 0.25 \cdot X_\tau \]

D5.C.6 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Depending on the model being designed, you may have to change some or all of the parameter default values. Some parameters are unit dependent and when altered, the n setting must be compatible with the active “unit” specification.

Table 7C.4 lists all the relevant EC3 parameters together with description and default values.

**Table 145: Steel Design Parameters EC3 EN**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALH</td>
<td>0.5</td>
<td>The ratio of the distance of the point torque (from the start of the member) to the length of the member. The default value of 0.5 represents torque acting at the mid-span of a symmetrically loaded member. Values can range from 0 to 1.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>ALPHA</td>
<td>1.0</td>
<td>Used to input a user defined value for the $\alpha$ factor in equation 6.41 for combined bending and axial force checks.</td>
</tr>
</tbody>
</table>
| BEAM           | 3             | Parameter to control the number of sections to checked along the length of a beam:  
1. Check at location of maximum $M_z$ along beam  
2. Check sections with end forces and forces at location of $\text{BEAM 1.0}$ check.  
3. Check at every $1/13\text{th}$ point along the beam and report the maximum |
| BETA           | 1.0           | Used to input a user defined value for the $\beta$ factor in equation 6.41 for combined bending and axial force checks. |
| C1             | 1.132         | Corresponds to the $C_1$ factor to be used to calculate Elastic critical moment $M_{cr}$ as per Clause 6.3.2.2 |
| C2             | 0.459         | Corresponds to the $C_2$ factor to be used to calculate Elastic critical moment $M_{cr}$ as per Clause 6.3.2.2 |
| C3             | 0             | Corresponds to the $C_3$ factor to be used to calculate Elastic critical moment $M_{cr}$ as per Clause 6.3.2.2 |
| CAN            | 0             | Member will be considered as a cantilever type member for deflection checks.  
0. indicates that member will not be treated as a cantilever member  
1. indicates that the member will be treated as a cantilever member |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CMM</td>
<td>1.0</td>
<td>Indicates type of loading and support conditions on member. Used to calculate the C1, C2, and C3 factors to be used in the $M_{cr}$ calculations. Can take a value from 1 to 8. Refer to Table 7C.3 for more information on its use.</td>
</tr>
<tr>
<td>CMN</td>
<td>1.0</td>
<td>Indicates the level of End-Restraint. 1.0 = No fixity 0.5 = Full fixity 0.7 = One end free and other end fixed</td>
</tr>
<tr>
<td>CMT</td>
<td>1</td>
<td>Used to indicate the loading and support condition for torsion (ref. SCI publication P-057). Can take a value of 1-7. The values correspond to the various cases defined in section 6 and App. B of SCI-P-057. Refer to Table 7C.4 for more information</td>
</tr>
<tr>
<td>DFF</td>
<td>0 (Mandatory for deflection check, TRACK 4.0)</td>
<td>&quot;Deflection Length&quot; / Max. allowable local deflection See Note 1d below.</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot;. See Note 1 below.</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot;. See Note 1 below.</td>
</tr>
<tr>
<td>DMAX</td>
<td>100.0 cm</td>
<td>Maximum allowable depth for the member.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0</td>
<td>Minimum required depth for the member.</td>
</tr>
<tr>
<td>EFT</td>
<td>Member Length</td>
<td>Effective length for torsion. A value of 0 defaults to the member length.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| ELB            | 0             | Used to specify the method for combined axial load + bending checks  
0. Uses Cl. 6.2.9 of EN 1993-1-1:2005  
1. Uses Cl. 6.2.1(7) - Eqn. 6.2 of EN 1993-1-1:2005 |
| ESTIFF         | 0             | (For use with the Dutch NA only) Method for checking columns forming part of (non)/butressed framework:  
0. Checks per Cl 12.3.1.2.3 of NEN 6770: Section 1  
1. Checks per Cl 12.3.1.2.3 of NEN 6770: Section 2  
Refer to D5.D.3.8 Clause 6.33 – Uniform members in bending and axial compression (on page 1802) for additional description on this parameter. |
| FAB            | 3             | Used to specify the fabrication class to be used to check for slender (Class 4) CHS/pipe sections (EN 1993-1-6:2007)  
1. Class A – Excellent  
2. Class B – High  
3. Class C – Normal |
<p>| FU             | 0             | Ultimate tensile strength of steel. |
| GM0            | 1.0           | Corresponds to the $\gamma_{m0}$ factor in EN 1993-1-1:2005 |
| GM1            | 1.0           | Corresponds to the $\gamma_{m1}$ factor in EN 1993-1-1:2005 |
| GM2            | 1.25          | Corresponds to the $\gamma_{m2}$ factor in EN 1993-1-1:2005 |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GST</td>
<td>0</td>
<td>Used to specify the section type to be used for designing a “General Section” from the user table. The member will be considered as the specified type with the user defined properties. The available options and corresponding values are as below:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. I-Section</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Single Channel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Rectangular Hollow Section</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. Circular Hollow Section</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4. Angle Section</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5. Tee Section</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Note: This parameter will be ignored if it has been assigned to any section other than a General Section.</td>
</tr>
<tr>
<td>KC</td>
<td>1.0</td>
<td>Corresponds to the correction factor as per Table 6.6 of EN 1993-1-1:2005. Program will calculate kc automatically if this parameter is set to 0.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Note: For the British, Singapore, &amp; Polish NAs, kc will be calculated as given in the NA by default.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K factor in local y axis. Used to calculate the effective length for slenderness and buckling calculations.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K factor in local z axis. Used to calculate the effective length for slenderness and buckling calculations.</td>
</tr>
<tr>
<td>LEG</td>
<td>0</td>
<td>Slenderness values for angles as determined from BS 5950-2000 Table 25. Refer to D3.B.6 Design Parameters (on page 1661)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LVV</td>
<td>Max. value of Lyy</td>
<td>Leg length for Lvw (length about v-v axis of single angle section), as per Lyy. Used for slenderness calculations.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Compression length in local y axis, Slenderness ratio = ((K_Y)*(L_Y)/(R_{yy}))</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Compression length in local z axis, Slenderness ratio = ((K_Z)*(L_Z)/(R_{zz}))</td>
</tr>
</tbody>
</table>
| MTH            | 0             | Used to select the clause to be used to calculate the LTB reduction factor, \(\chi_{LT}\). The available options and corresponding values are as below:  
  0. Use default method based on section type (default)  
  1. Use Cl.6.3.2.2  
  2. Use Cl.6.3.2.3  
  
By default, the program will use Cl 6.3.2.3 for rolled & built-up I-sections and Cl. 6.3.2.2 for all other sections. If, however, the specified National Annex expands on Cl. 6.3.2.3 to include other section types (e.g., the UK NA), the program will use Cl. 6.3.2.3 by default for that particular section type.  

| MU             | 0             | To be used with CMM values of 7 and 8. See Table 7C.3.  
  
**Note:** Currently valid only with the French & Belgian NAs. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NA</td>
<td>0</td>
<td>Choice of National Annex to be used for EC3 design. Refer to [D5.D. European Codes - National Annexes to Eurocode 3 <a href="en">EN 1993-1-1:2005</a>](en) (on page 1792) for values allowed for this parameter. (Refer to <a href="en">D5.C.1 General Description</a> (on page 1755) for more information)</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net tension factor for tension capacity calculation.</td>
</tr>
<tr>
<td>PLG</td>
<td>0</td>
<td>To be used to determine whether to include the additional interaction checks as per CL. NA.20(2) and NA.20(3) of the <a href="en">Polish National Annex</a> (on page 1826).</td>
</tr>
<tr>
<td>Note: This parameter will be applicable only to the Polish NA</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PY</td>
<td>Yield Strength</td>
<td>The yield strength default value is set based on the default value of the SGR parameter.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1</td>
<td>Permissible ratio of loading to capacity.</td>
</tr>
<tr>
<td>SBLT</td>
<td>0.0</td>
<td>Indicates if the section is rolled or built-up.</td>
</tr>
<tr>
<td>0.0 = Rolled</td>
<td>1.0 = Built-up</td>
<td></td>
</tr>
<tr>
<td>2.0 = Cold-formed (uses the appropriate buckling curve from Table 6.2)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SGR</td>
<td>0</td>
<td>Steel grade as in Table 3.1: EN 1993-1-1: 2005:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0 - indicates S 235 grade steel - EN10025-2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 - indicates S 275 grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.0 - indicates S 355 grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3.0 - indicates S 450 grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4.0 - indicates S 275 N/NL grade steel - EN10025-3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5.0 - indicates S 355 N/NL grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6.0 - indicates S 420 N/NL grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7.0 - indicates S 460 N/NL grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>8.0 - indicates S 275 M/ML grade steel - EN10025-4</td>
</tr>
<tr>
<td></td>
<td></td>
<td>9.0 - indicates S 355 M/ML grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>10.0 - indicates S 420 M/ML grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>11.0 – indicates S 460 M/ML grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>12.0 – indicates S 235 W grade steel - EN10025-5</td>
</tr>
<tr>
<td></td>
<td></td>
<td>13.0 – indicates S 355 W grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>14.0 – indicates S 460 Q/QL/QL1 grade steel - EN10025-6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>15.0 – indicates S 235 H grade steel - EN10210-1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>16.0 – indicates S 275 H grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>17.0 – indicates S 355 H grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>18.0 – indicates S 275 NH/NLH grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>19.0 – indicates S 355 NH/NLH grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>20.0 – indicates S 420 NH/NLH grade steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>21.0 – indicates S 460 NH/NLH grade steel</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>22.0</td>
<td></td>
<td>22.0 – indicates S 235 H grade steel - EN10219-1</td>
</tr>
<tr>
<td>23.0</td>
<td></td>
<td>23.0 – indicates S 275 H grade steel</td>
</tr>
<tr>
<td>24.0</td>
<td></td>
<td>24.0 – indicates S 355 H grade steel</td>
</tr>
<tr>
<td>25.0</td>
<td></td>
<td>25.0 – indicates S 275 NH/NLH grade steel</td>
</tr>
<tr>
<td>26.0</td>
<td></td>
<td>26.0 – indicates S 355 NH/NLH grade steel</td>
</tr>
<tr>
<td>27.0</td>
<td></td>
<td>27.0 – indicates S 460 NH/NLH grade steel</td>
</tr>
<tr>
<td>28.0</td>
<td></td>
<td>28.0 – indicates S 275 MH/MLH grade steel</td>
</tr>
<tr>
<td>29.0</td>
<td></td>
<td>29.0 – indicates S 355 MH/MLH grade steel</td>
</tr>
<tr>
<td>30.0</td>
<td></td>
<td>30.0 – indicates S 420 MH/MLH grade steel</td>
</tr>
<tr>
<td>31.0</td>
<td></td>
<td>31.0 – indicates S 460 MH/MLH grade steel</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td></td>
<td>As EN 1993-1-1:2005 does not provide a buckling curve in table 6.2 for grade S 450 steel (in Table 3.1 of EN 1993-1-1:2005), the program will use the same buckling curves as for grade S 460 when calculating the buckling resistance as per clause 6.3.</td>
</tr>
<tr>
<td>STIFF</td>
<td></td>
<td>Member Length or depth of beam, whichever is lesser</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Distance between transverse stiffener plates, used to prevent web shear buckling. If not specified or if a value of 0 is provided, the program will assume the web is unstiffened.</td>
</tr>
<tr>
<td>TOM</td>
<td>0</td>
<td>Total torsion for design used for torsion checks. Can be used to override the total torsional moment to be used for member design.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>TORSION</td>
<td>0</td>
<td>Method to be used for a specific member or group of members:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Perform basic torsion checks if member is subject to torsion.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Perform basic stress check (Ignore warping effects).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Perform detailed checks (including warping effects).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. Ignore all torsion checks</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> For options 1 or 2, the program will perform the torsion related checked even if torsional moment is absent and will use a value of zero for the torsional moment.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>Specify level of detail in output.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Summary of results only.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Summary with member capacities.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Detailed results.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4. Deflection check results only.</td>
</tr>
<tr>
<td>UNF</td>
<td>1</td>
<td>Unsupported length as a fraction of the actual member length.</td>
</tr>
<tr>
<td>UNL</td>
<td>Member Length</td>
<td>Unrestrained length of member used in calculating the lateral-torsional resistance moment of the member.</td>
</tr>
<tr>
<td>ZG</td>
<td>+Section Depth/2</td>
<td>Distance of transverse load from shear center. Used to calculate $M_{cr}$.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> For Tee sections, ZG will have a default value of (+Flange thickness/2)</td>
</tr>
</tbody>
</table>

**Notes**

1. **CAN, DJ1, and DJ2 – Deflection**

   a. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. Refer to [TR.44 Printing Section Displacements for Members](on page 2846) for details on local displacement.
If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1, DZ1) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at DJ2 or the end node of the member.

Compute \( \Delta = \sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2} \)

Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

Then, if CAN is specified a value 1, \( \text{dff} = L/\Delta \)

**Ratio due to deflection** = \( \text{DFF}/\text{dff} \)

b. If CAN = 0, deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.

\[ D = \text{Maximum local deflection for members 1, 2, and 3.} \]

<table>
<thead>
<tr>
<th>PARAMETERS</th>
</tr>
</thead>
<tbody>
<tr>
<td>DFF 300. ALL</td>
</tr>
<tr>
<td>DJ1 1 ALL</td>
</tr>
<tr>
<td>DJ2 4 ALL</td>
</tr>
</tbody>
</table>

c. If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

d. It is important to note that unless a DFF value is specified, STAAD will not perform a deflection check. This is in accordance with the fact that there is no default value for DFF (see Table 2B.1).

e. The above parameters may be used in conjunction with other available parameters for steel design.

2. **CMM Parameter**

The values of CMM for various loading and support conditions are as given below:

**Table 146: Values for the CMM Parameter**

<table>
<thead>
<tr>
<th>CMM Value</th>
<th>Loading and Support Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td><img src="image1.png" alt="Image 1" /></td>
</tr>
<tr>
<td>2</td>
<td><img src="image2.png" alt="Image 2" /></td>
</tr>
</tbody>
</table>
### CMM Value

<table>
<thead>
<tr>
<th>CMM Value</th>
<th>Loading and Support Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td><img src="image1" alt="Diagram" /></td>
</tr>
<tr>
<td>4</td>
<td><img src="image2" alt="Diagram" /></td>
</tr>
<tr>
<td>5</td>
<td><img src="image3" alt="Diagram" /></td>
</tr>
<tr>
<td>6</td>
<td><img src="image4" alt="Diagram" /></td>
</tr>
<tr>
<td>7</td>
<td><img src="image5" alt="Diagram" /></td>
</tr>
<tr>
<td>8</td>
<td><img src="image6" alt="Diagram" /></td>
</tr>
</tbody>
</table>

3. Checking beam deflection

With the TRACK parameter set to 4, the members included in a BEAM CHECK command will be checked for the local axis deflection rather than for the stress capacity using the current LOAD LIST.

If both stress capacity and deflection checks are required, then 2 parameter blocks with code checks are required, one with a TRACK 4 command and one with a TRACK 0, 1 or 2, thus:

```plaintext
LOAD LIST 1 TO 10
PARAMETER 1
CODE EN 1993
TRACK 2 ALL
CODE CHECK MEMBER 1
***************************
LOAD LIST 100 TO 110
PARAMETER 2
TRACK 4 ALL
DFF 300 MEMB 1
DJ1 1 MEMB 1
DJ2 4 MEMB 1
CHECK CODE MEMB 1
```
Note: While both sets of code checks will be reported in the output file, only the last code check results are reported in the STAAD.Pro graphical interface.

4. CMT Parameter

The values of CMT for various loading and support conditions are as given below:

Table 147: Loading and Support Conditions represented by CMT Parameter Values

<table>
<thead>
<tr>
<th>CMT Value</th>
<th>Description</th>
<th>Diagram</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(Default): Concentrated Torque at Ends. Ends Torsion fixed and Warping fixed</td>
<td><img src="image1" alt="Diagram" /></td>
</tr>
<tr>
<td>2</td>
<td>Concentrated Torque along length of member. Ends Torsion fixed and Warping free</td>
<td><img src="image2" alt="Diagram" /></td>
</tr>
<tr>
<td>3</td>
<td>Concentrated Torque along length of member. Ends Torsion fixed and Warping fixed</td>
<td><img src="image3" alt="Diagram" /></td>
</tr>
<tr>
<td>4</td>
<td>Uniform Torque in member. Ends Torsion fixed and Warping free</td>
<td><img src="image4" alt="Diagram" /></td>
</tr>
<tr>
<td>5</td>
<td>Uniform Torque in member. Ends Torsion fixed and Warping fixed</td>
<td><img src="image5" alt="Diagram" /></td>
</tr>
<tr>
<td>6</td>
<td>Concentrated Torque in cantilever. End Torsion fixed and Warping fixed</td>
<td><img src="image6" alt="Diagram" /></td>
</tr>
<tr>
<td>7</td>
<td>Uniform Torque in cantilever. End Torsion fixed and Warping fixed</td>
<td><img src="image7" alt="Diagram" /></td>
</tr>
</tbody>
</table>
Note: For CMT = 2 and CMT = 3, you have the option of specifying the distance at which the concentrated torque acts, measured from the start of the member. This can be done by using the ALH design parameter. The ALH parameter indicates the ratio of the distance of the point torque (from the start of the member) to the length of the member. This parameter will have a default value of 0.5 (i.e., the torque acts at the center of the span) and will accept values ranging from 0 to 1.

Note: The GB1 parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005. Hence any legacy files that use GB1 parameter will indicate an error message and you will be required to substitute GB1 with GM1, in accordance with EN 1993-1-1:2005.

Related Links
- **AD.2007-11.3.9 Eurocode 3 Steel Grades** (on page 117)
- **AD.2007-03.2.5 Updated and Additional Standard Steel Grades in Eurocode 3** (on page 226)

### D5.C.7 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate. The adequacy is checked as per EN 1993-1-1:2005 and a corresponding National Annex (if specified). Code checking is done using the forces and moments at specific sections of the members.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks; the critical condition; the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member of forces in the member where the critical condition occurs).

Code checking can be done with any type of steel section listed in **Section 2B.4** (on page 1657) or any of the user defined sections as described in **G.6.3 User-Provided Steel Table** (on page 2325), with the exception of ISECTION. ISECTION has been currently excluded since the option of Tapered section design is currently not supported in the EC3 module. The EC3 (EN 1993) design module does not consider these sections or PRISMATIC sections in its design process.

**Note:** Checks for slender sections to EN 1993-1-1 are limited to I-SECTIONS, TEE, SINGLE CHANNEL, SINGLE ANGLE and CIRCULAR & RECTANGULAR HOLLOW SECTIONS.

Code checking for **GENERAL** sections can be also done using the EN1993 module. The program will design **GENERAL** sections as I sections by default. However, you are given the option to choose a "section type" to be considered while designing the member. Refer to the description of the GST design parameter in **Section 7C.6** (on page 1776) for details.

### D5.C.8 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN, which limits the maximum and minimum depth of the members.

Member selection can be performed with all the types of steel sections with the same limitations as defined in **Section 7C.7** (on page 1789).
Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table.

Member selection cannot be performed on members whose section properties are input as prismatic or as the limitations specified in Section 7C.7 (on page 1789).

### D5.C.9 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

| **MEMBER** | refers to the member number for which the design is performed. |
| **TABLE** | refers to steel section name, which has been checked against the steel code or has been selected. |
| **RESULTS** | prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member. |
| **CRITICAL COND** | refers to the clause in EN 1993-1-1:2005 code which governs the design. |
| **RATIO** | prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed. |
| **LOADING** | provides the load case number, which governed the design. |
| **FX, MY, and MZ** | provide the axial force, moment in local Y-axis and the moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases. |
| **LOCATION** | specifies the actual distance from the start of the member to the section where design forces govern. |

**Note:** For a TRACK 2 output, the module will also report all the relevant clause checks that have been performed and will also indicate the critical ratio and the load case that caused the critical ratio as well as the corresponding forces that were used for the respective checks. A TRACK 2 output will also include the various design data used for the calculations such as the section modulii, section class, section capacity etc.

If an NA parameter (other than 0) has been specified and if the particular National Annex requires additional checks outside those specified in EN 1993-1-1:2005 (e.g., The Dutch National Annex), the respective NA clauses and any associated code clauses will be listed along with the critical ratios and the forces that were used for these clause checks.

Documentation notes appear in red.

**Note:** The results and output follow the axis convention as described in Section 7C.1.3

<table>
<thead>
<tr>
<th>Code title &amp; version</th>
<th>STAAD.PRO CODE CHECKING - BS EN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1993-1-1:2005</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>National Annex used, if any</th>
<th>NATIONAL ANNEX - NA to BS EN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1993-1-1:2005</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Design engine version</th>
<th>PROGRAM CODE REVISION V1.9 BS_EC3_2005/1</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALL UNITS ARE -</td>
<td></td>
</tr>
<tr>
<td>METE (UNLESS OTHERWISE NOTED)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT</th>
<th>CRITICAL COND</th>
<th>RATIO</th>
<th>LOADING</th>
<th>FX</th>
<th>MY</th>
<th>MZ</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Note:** For a TRACK 2 output, the module will also report all the relevant clause checks that have been performed and will also indicate the critical ratio and the load case that caused the critical ratio as well as the corresponding forces that were used for the respective checks. A TRACK 2 output will also include the various design data used for the calculations such as the section modulii, section class, section capacity etc.

If an NA parameter (other than 0) has been specified and if the particular National Annex requires additional checks outside those specified in EN 1993-1-1:2005 (e.g., The Dutch National Annex), the respective NA clauses and any associated code clauses will be listed along with the critical ratios and the forces that were used for these clause checks.

Documentation notes appear in red.
### Member number, section profile & table

1 ST HD320X127 (EUROPEAN SECTIONS)

- Design status, critical code clause, & critical ratio:
  - PASS
  - Design Code: EC-6.3.3-662
  - Critical ratio: 0.045
  - Section forces & critical section location:
    - FX: 25.00
    - MY: 5.00
    - MZ: 0.00

---

### Material Data

- Grade of steel: USER
- Modulus of elasticity: 205 kN/mm²
- Design Strength (py): 275 N/mm²

### Section Properties (units - cm)

- Member Length: 500.00
- Gross Area: 161.30
- Net Area: 161.30

"z-axis" here refers to bending about Z-Z (when Y is Up), whereas EC3 uses the Y-Y axis convention (on page 1755).

<table>
<thead>
<tr>
<th></th>
<th>z-axis</th>
<th>y-axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment of inertia</td>
<td>30820.004</td>
<td>9239.001</td>
</tr>
<tr>
<td>Plastic modulus</td>
<td>2149.000</td>
<td>939.100</td>
</tr>
<tr>
<td>Elastic modulus</td>
<td>1926.250</td>
<td>615.933</td>
</tr>
<tr>
<td>Shear Area</td>
<td>81.998</td>
<td>51.728</td>
</tr>
<tr>
<td>Radius of gyration</td>
<td>13.823</td>
<td>7.568</td>
</tr>
<tr>
<td>Effective Length</td>
<td>500.000</td>
<td>500.000</td>
</tr>
</tbody>
</table>

### Design Data (units - kN,m) EUROCODE NO.3 /2005

- Section class as per Table 5.2
- Section Class: CLASS 1
- Max. cross section capacity (A · f_y/GM₀)
  - Squash Load: 4435.75
- Axial force/Squash load: 0.006
- Partial safety factors used
  - GM0: 1.00
  - GM1: 1.00
  - GM2: 1.10

<table>
<thead>
<tr>
<th></th>
<th>z-axis</th>
<th>y-axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio (KL/r)</td>
<td>36.2</td>
<td>66.1</td>
</tr>
<tr>
<td>Compression Capacity</td>
<td>4078.2</td>
<td>3045.5</td>
</tr>
<tr>
<td>Tension Capacity</td>
<td>4435.8</td>
<td>4435.8</td>
</tr>
<tr>
<td>Moment Capacity</td>
<td>591.0</td>
<td>258.3</td>
</tr>
<tr>
<td>Reduced Moment Capacity</td>
<td>591.0</td>
<td>258.3</td>
</tr>
<tr>
<td>Shear Capacity</td>
<td>1301.9</td>
<td>821.3</td>
</tr>
</tbody>
</table>

### Buckling Calculations (units - kN,m)

- Lateral Torsional Buckling Moment: MB = 591.0
- Factor C1 used in Mcr calculations and End restraint factor (corresponds to the CMN design parameters)
  - co-efficients C1 & K: C1 = 2.578, K = 1.0, Effective Length = 5000

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic Critical Moment for LTB,</td>
<td>Mcr = 1541.5</td>
</tr>
<tr>
<td>Critical Load For Torsional Buckling,</td>
<td>NcrT = 13898.0</td>
</tr>
<tr>
<td>Critical Load For Torsional-Flexural Buckling,</td>
<td>NcrTF = 13898.0</td>
</tr>
</tbody>
</table>

### All Units Are - KN METE (UNLESS OTHERWISE NOTED)

<table>
<thead>
<tr>
<th>MEMBER STYLE</th>
<th>TABLE</th>
<th>RESULT/ FX</th>
<th>CRITICAL COND/ MY</th>
<th>RATIO/ MZ</th>
<th>LOADING/ LOCATION</th>
</tr>
</thead>
</table>

### Critical Loads for Each Clause Check (units- kN,m):
Max. ratio, loadcase, & section forces for each clause check

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>EC-6.3.1.1</td>
<td>0.008</td>
<td>1</td>
<td>25.0</td>
<td>0.0</td>
<td>0.0</td>
<td>-10.0</td>
<td>5.0</td>
</tr>
<tr>
<td>EC-6.2.9.1</td>
<td>0.020</td>
<td>1</td>
<td>25.0</td>
<td>0.0</td>
<td>0.0</td>
<td>-10.0</td>
<td>5.0</td>
</tr>
<tr>
<td>EC-6.3.3-661</td>
<td>0.035</td>
<td>1</td>
<td>25.0</td>
<td>0.0</td>
<td>0.0</td>
<td>-10.0</td>
<td>5.0</td>
</tr>
<tr>
<td>EC-6.3.3-662</td>
<td>0.045</td>
<td>1</td>
<td>25.0</td>
<td>0.0</td>
<td>0.0</td>
<td>-10.0</td>
<td>5.0</td>
</tr>
<tr>
<td>EC-6.3.2 LTB</td>
<td>0.017</td>
<td>1</td>
<td>25.0</td>
<td>0.0</td>
<td>0.0</td>
<td>-10.0</td>
<td>5.0</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.

*************** END OF TABULATED RESULT OF DESIGN ***************


A number of countries that have signed up to the replace their current steel design standards with the Eurocode, EN 1993-1-1:2005, known commonly as Eurocode 3, have published their National Annex documents. These documents make small changes to the base document and STAAD.Pro has been updated to incorporate some of these National Annex documents.

The parameter NA sets the default material gamma factors and any additional changes outlined in the country specific National Annex such as specific equations or methods. These are described for each National Annex document in the following sections.

The output file printout has been updated to indicate which National Annex (if any) has been used in a code check / select process (For all TRACK settings).

D5.D.1 General Format

The format of the EN 1993-1-1:2005 National Annex is as follows:

<table>
<thead>
<tr>
<th>CODE EN 1993</th>
</tr>
</thead>
<tbody>
<tr>
<td>NA f1</td>
</tr>
<tr>
<td>{code parameters}</td>
</tr>
</tbody>
</table>

Refer to D5.C.6 Design Parameters (on page 1776) (on page 1776)

Where:

f1 represents the number designation for a specific country's National Annex:

<table>
<thead>
<tr>
<th>NA Value</th>
<th>Country</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>None — Uses the base EN 1993-1-1:2005 code, with no national annex changes or additions. The default values specified in En 1993-1-1:2005 will be used for the partial safety factors and various parameter values where applicable (default).</td>
</tr>
</tbody>
</table>

---

STAAD.Pro 1792 User Manual
<table>
<thead>
<tr>
<th>NA Value</th>
<th>Country</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>United Kingdom ([British NA](on page 1804)) — Uses the BS EN 1993-1-1:2005 version of Eurocode 3 along with the UK National Annex.</td>
</tr>
<tr>
<td>2</td>
<td>Netherlands ([Dutch NA](on page 1794)) — Uses the NEN EN 1993-1-1:2005 version of the code. The Dutch National Annex [NEN-EN 1993-1-1/NB] has been added in this module. Please note that the Dutch National requires additional checks as per NEN 6770 and NEN 6771 which will also be performed during design checks with this parameter value.</td>
</tr>
<tr>
<td>3</td>
<td>Norway ([Norwegian NA](on page 1803)) — Uses the NS-EN 1993-1-1:2005 version of the code. The Norwegian National Annexe [NS-EN 1993-1-1:2005/Na 2008] has been added to this implementation.</td>
</tr>
<tr>
<td>4</td>
<td>France ([French NA](on page 1813)) — Uses the Annexe Nationale a la NF EN 1993-1-1:2005 version of the code along with the French National Annex.</td>
</tr>
<tr>
<td>5</td>
<td>Finland ([Finnish NA](on page 1820)) - Uses the SFS EN 1993-1-1:2005 version of Eurocode 3 along with the Finnish National Annex.</td>
</tr>
<tr>
<td>6</td>
<td>Poland ([Polish NA](on page 1826)) - Uses the PN EN 1993-1-1:2005 version of Eurocode 3 along with the Polish National Annex.</td>
</tr>
<tr>
<td>7</td>
<td>Singapore ([Singaporean NA](on page 1831)) - Uses the SS EN 1993-1-1:2005 version of Eurocode 3 along with the Singaporean National Annex.</td>
</tr>
<tr>
<td>8</td>
<td>Belgium ([Belgian NA](on page 1838)) - Uses the NBN EN 1993-1-1:2005 version of Eurocode 3 along with the Belgian National Annex.</td>
</tr>
<tr>
<td>9</td>
<td>Malaysian ([Malaysian NA](on page 1844)) - Uses the MS EN 1993-1-1:2005 version of Eurocode 3 along with the Malaysian National Annex.</td>
</tr>
<tr>
<td>10</td>
<td>German ([German NA](on page 1849)) - Uses the DIN EN 1993-1-1:2005 version of Eurocode 3 along with the German National Annex.</td>
</tr>
<tr>
<td>11</td>
<td>Swedish ([Swedish NA](on page 1853)) - Uses the BFS EN 1993-1-1:2005 version of Eurocode 3 along with the Swedish National Annex.</td>
</tr>
</tbody>
</table>
D5.D.2 Specifying the design engine to use a national annex

Use the following procedure to include additional check specified by a National Annex:

Batch steel design is performed in the Analytical Modeling workflow.

1. On the Analysis and Design ribbon tab, select Steel in the Design gallery.
   The Steel Design - Whole Structure dialog box opens.
3. Click Define Parameters…
   The Design Parameters dialog box opens.
4. Select the NA parameter in the list box.
5. Select the option corresponding to the National Annex document you want to use.
6. Click Add.

   This will insert the following commands into the STAAD input file:

   ```plaintext
   CODE EN 1993-1-1:2005
   NA n
   ```

Refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1754) for additional information on steel design per EC3.

A design performed to the new Eurocode 3 National Annex is displayed in the output file (*.ANL) with the following header, in addition to the base EC3 output.

D5.D.3 Dutch National Annex to EC3


Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the D-NA.

**Note:** Clause 6.3.2.4 deals with a simplified assessment method for beams. STAAD.Pro only uses the more accurate method (6.3.2.2 and 6.3.2.3 in EC-3) and therefore this section is ignored.

The local axis convention in the Dutch codes is: Y – major axis & Z – minor axis (as opposed to the convention followed in STAAD.Pro).
D5.D.3.2 Clause 6.1 – General

The partial safety factors will use the following values:

- Resistance of cross-sections, $\gamma_{M0} = 1.0$
- Resistance of members to instability, $\gamma_{M1} = 1.0$
- Resistance of cross sections to tension, $\gamma_{M2} = 1.25$

The design function in STAAD.Pro sets these values as the default values for the D-NA (NA 3 is specified).

**Note:** You can change these values through the GM0, GM1, & GM2 design parameters. Refer to D5.C.6 Design Parameters (on page 1776)

D5.D.3.3 Clause 6.2.8 – Bending and shear

The D-NA requires the implementation of causes 11.3.1.1 and 11.3.1.3 of NEN 6770.

Clause 11.3.1.1 (NEN 6770): Class 1 and Class 2 I-section profiles

Class 1 and class 2 I section profiles must satisfy the interaction formulae given in tables 10 & 11 of NEN 6770.

Table 10 Provides interaction checks for bending about the major axis (All necessary terms and formulae are described below):

1. If $V_{z,s;d} \leq 0.5 \cdot V_{z,pl;d}$ and $N_{s;d} \leq 0.5 \cdot a_1 \cdot N_{pl;d}$, check equation 11.3.1
2. If $V_{z,s;d} \leq 0.5 \cdot V_{z,pl;d}$ and $N_{s;d} > 0.5 \cdot a_1 \cdot N_{pl;d}$, check equation 11.3.2
3. If $V_{z,s;d} > 0.5 \cdot V_{z,pl;d}$ and $N_{s;d} \leq 0.5 \cdot a_2 \cdot N_{v,u;d}$, check equation 11.3-3
4. If \( V_{z;sd} > 0.5 \cdot V_{z;pl;d} \) and \( N_{s;d} > 0.5 \cdot a_2 \cdot N_{v;ud;d} \), check equation 11.3-4

where

\[
\begin{align*}
V_{z;sd} &= \text{Actual Shear force in the section along Z-axis} \\
V_{z;pl;d} &= \text{Shear capacity of section along Z-axis} \\
&= A_w \cdot f_{yd}/\sqrt{3} \\
f_{yd} &= \text{yield stress} \\
A_w &= A \cdot 2 (bf - tw - 2r) tf \\
N_{s;d} &= \text{Axial force in the section} \\
N_{pl;d} &= \text{Axial capacity of section} = A \cdot f_{yd} \\
M_{y;sd} &= \text{Bending moment about major axis} \\
M_{y;pl;d} &= \text{Plastic moment capacity of section} = f_{yd} \cdot W_{pl} \\
W_{y;pl} &= \text{Plastic section modulus} \\
\alpha_1 &= \min(A - 2bf \cdot tf)/A, 0.5) \text{- used in tables 10 & 11} \\
\alpha_2 &= \text{see eqn 11.3-10 \text{- used in tables 10 & 11}} \\
M_{v;y;ud} &= \text{see eqn 11.3.12} \\
N_{v;u;d} &= \text{see eqn 11.3-13} \\
\end{align*}
\]

Table 11: Provides interaction formulae for bending about the minor axis

1. If \( V_{y;sd} \leq 0.25 \cdot V_{y;pl;d} \) and \( N_{s;d} \leq 1.0 \cdot \alpha_1 \cdot N_{pl;d} \), check equation 11.3-5
2. If \( V_{y;sd} \leq 0.25 \cdot V_{y;pl;d} \) and \( N_{s;d} > 1.0 \cdot \alpha_1 \cdot N_{pl;d} \), check equation 11.3-6
3. If \( V_{y;sd} > 0.25 \cdot V_{y;pl;d} \) and \( N_{s;d} \leq 1.0 \cdot \alpha_1 \cdot N_{v;u;d} \), check equation 11.3-7
4. If \( V_{y;sd} > 0.25 \cdot V_{y;pl;d} \) and \( N_{s;d} > 1.0 \cdot \alpha_1 \cdot N_{v;u;d} \), check equation 11.3-8

where

\[
\begin{align*}
V_{y;sd} &= \text{Actual Shear force in the section along Y-axis} \\
V_{y;pl;d} &= \text{Shear capacity of section along Y-axis} \\
&= 2bt \cdot f_{yd}/\sqrt{3} \\
M_{v;z;ud} &= q \cdot M_{z;pl;d} = q \cdot f_{yd} \cdot W_{pl;z;d} \\
\end{align*}
\]
Clause 11.3.1.3 (NEN 6770): Class 1 and Class 2 Square and rectangular hollow sections

This clause requires class 1 and class 2 square and rectangular tube profiles to satisfy the interaction equations in Table 13.

1. If $V_{z;sd} \leq 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} \leq 0.5 \cdot a_3 \cdot N_{pl;d}$ check equation 11.3.22
2. If $V_{z;sd} \leq 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} > 0.5 \cdot a_3 \cdot N_{pl;d}$ check equation 11.3.23
3. If $V_{z;sd} > 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} \leq 0.5 \cdot a_4 \cdot N_{v;u;d}$ check equation 11.3-24
4. If $V_{z;sd} > 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} > 0.5 \cdot a_4 \cdot N_{v;u;d}$ check equation 11.3-25

where

- $V_{z;sd}$ = Actual Shear force in the section along Z-axis
- $V_{z;pl;d}$ = Shear capacity of section along Z-axis
- $b$ = breadth of section
- $h$ = height of section
- $A$ = area of section
- $V_{z;pl;d}$ = $V_z;cl;d = \frac{h}{b + h} \cdot A \cdot f_{y;d} \sqrt{3}$
- $a_3 = min\{ (A \cdot 2 \cdot b \cdot t)/A \text{ or } 0.5\}$
- $a_4 = \text{ from equation 11.3.27}$

D5.D.3.4 Clause 6.2.10 – Bending shear and axial force

Requires the implementation of clauses 11.3.1.1 to 11.3.1.3 and 11.3.2.1 to 11.3.2.3 of NEN 6770 and clause 11.3 of NEN 6771

Clause 11.3.1.1 (NEN 6770) and Clause 11.3.1.3 (NEN 6770)

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1795)

Clause 11.3.1.2 (NEN 6770): Class 1 and class 2 circular hollow (CHS) profiles

Class 1 and class 2 sections with circular hollow profiles should satisfy the interaction equations given in table 12.

- Check #1 - If $V_{z;sd} \leq 0.25 \cdot V_{z;pl;d}$ check equation 11.3.17
- Check #2 - If $V_{z;sd} > 0.25 \cdot V_{z;pl;d}$ check equation 11.3.18.

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1795) of this document for equations to derive $V_{z;sd}$

$V_{z;pl;d} = \frac{2A}{\pi} f_{y;d} \sqrt{3}$

See equations 11.3-19 and 11.3-20 to evaluate $M_{v;y;u;d}$ and $N_{v;u;d}$.

To check for these conditions about the y axis, substitute the index “z” in the above equations with “y” (should be the same of CHS sections).
Clause 11.3.2 (NEN 6770)

Section 11.3.2 in general deals with biaxial bending with axial force and shear. The general condition to be satisfied in this case is given by equation 11.3-31 of NEN 6770

\[
\beta_0 \left( \frac{M_{y,s};d}{M_{N;v,y;u;d}} \right)^{a_1} + \beta_1 \left( \frac{M_{z,s};d}{M_{N;v,z;u;d}} \right)^{a_2} \leq 1
\]

Clause 11.3.2.1: Class 1 and class 2 I-sections with biaxial bending + shear + axial force

The formula to evaluate \( M;N;v;y;u;d \) and \( M;N;v;z;u;d \) are to be taken from tables 14 and 15 of NEN 6770 respectively.

Checks for table 14:

1. Check #1 – If \( V_z;s;d \leq 0.5 \) \( V_z;pl;d \) and \( N_s;d \leq 0.5 \times a_1 \times N_{pl;d} \) use equation 11.3.32
2. Check #2 – If \( V_z;s;d \leq 0.5 \) \( V_z;pl;d \) and \( N_s;d > 0.5 \times a_1 \times N_{pl;d} \) use equation 11.3.33
3. Check #3 – If \( V_z;s;d > 0.5 \) \( V_z;pl;d \) and \( N_s;d \leq 0.5 \times a_2 \times N_{v;u;d} \) use equation 11.3-34
4. Check #4 – If \( V_z;s;d > 0.5 \) \( V_z;pl;d \) and \( N_s;d > 0.5 \times a_2 \times N_{v;u;d} \) use equation 11.3-35

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1795) for equations to evaluate \( V_z;s;d \), \( M_{y;pl;d} \), \( N_{pl;d} \), \( M_{v;y;u;d} \), \( N_{v;u;d} \), \( a_1 \), \( a_2 \) and \( V_z;pl;d \).

Checks for table 15:

1. Check #1 – If \( V_y;s;d \leq 0.25 \) \( V_y;pl;d \) and \( N_s;d \leq 1.0 \times a_1 \times N_{pl;d} \) use equation 11.3.36
2. Check #2 – If \( V_y;s;d \leq 0.25 \) \( V_y;pl;d \) and \( N_s;d > 1.0 \times a_1 \times N_{pl;d} \) use equation 11.3.37
3. Check #3 – If \( V_y;s;d > 0.25 \) \( V_y;pl;d \) and \( N_s;d \leq 1.0 \times a_1 \times N_{v;u;d} \) check equation 11.3-38
4. Check #4 – If \( V_y;s;d > 0.25 \) \( V_y;pl;d \) and \( N_s;d > 1.0 \times a_1 \times N_{v;u;d} \) check equation 11.3-39

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1795) for equations to evaluate \( V_y;s;d \), \( M_{z;pl;d} \), \( N_{pl;d} \), \( M_{v;z;u;d} \), \( N_{v;u;d} \), \( a_1 \), \( a_2 \) and \( V_y;pl;d \).

See table 16 for \( \alpha_1, \alpha_2, \beta_0 \) and \( \beta_1 \) use in tables 14 and 15.

Clause 11.3.2.2: Class 1 and Class 2 Circular hollow tubes

The formula to evaluate \( M;N;v;y;u;d \) and \( M;N;v;z;u;d \) (to be used in equation 11.3-31, see description of clause 11.3.2 above) are to be taken from table 17 of NEN 6770.

1. Check #1 – If \( V_z;s;d \leq 0.25 \) \( V_z;pl;d \) use equation 11.3.44
2. Check #2 – If \( V_z;s;d > 0.25 \) \( V_z;pl;d \) use equation 11.3.45.

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1795) for equations to evaluate \( V_z;pl;d \), \( M_{y;pl;d} \), and \( N_{pl;d} \) use in equations 11.3.44 & 11.3.45.

For values to be used for \( \alpha_1, \alpha_2 \), \( \beta_1 \) and \( \beta_2 \) in this case refer to table 18 of NEN 6770.

Clause 11.3.2.3: Class 1 and Class 2 Rectangular and square hollow tubes

The formula to evaluate \( M;N;v;y;u;d \) and \( M;N;v;z;u;d \) (to be used in equation 11.3-31, see description of clause 11.3.2 above) are to be taken from table 19 of NEN 6770.

1. Check #1 – If \( V_z;s;d \leq 0.25 \) \( V_z;pl;d \) and \( N_s;d \leq 0.5 \times a_3 \times N_{pl;d} \) use equation 11.3-48
2. Check #2 – If \( V_z;s;d \leq 0.25 \) \( V_z;pl;d \) and \( N_s;d > 0.5 \times a_3 \times N_{pl;d} \) use equation 11.3.49
3. Check #3 – If \( V_z;s;d > 0.25 \) \( V_z;pl;d \) and \( N_s;d \leq 0.5 \times a_4 \times N_{v;u;d} \) use equation 11.3-50
4. Check #4 – If \( V_z;s;d > 0.25 \) \( V_z;pl;d \) and \( N_s;d > 0.5 \times a_4 \times N_{v;u;d} \) check equation 11.3-51
Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1795) for equations to evaluate $V_{z;pl;d}$, $M_{y;pl;d}$, $N_{pl;d}$, $M_{y;ud}$, $N_{y;ud}$, $a_3$, $a_4$ and $V_{z;pl;d}$ to be used in the above equations. For values to be used for $\alpha_1$, $\alpha_2$, $\beta_1$ and $\beta_2$ in this case refer to table 20 of NEN 6770.

To check for these conditions about the y axis, substitute the index "z" in the above equations with 'y'.

Clause 11.3 (NEN 6771)

In general, this section deals with Biaxial bending with axial force and shear for class 3 and class 4 sections.

Check for class 3 sections: For class 3 sections use the method in section 11.3 NEN 6770. For class 3 sections the methods and equations discussed above can be used with the "plastic section modulus" being substituted with the 'elastic modulus'.

Check for class 4 sections: Class 4 sections can be treated as class 3 sections if the effective section properties are used as given in clause 10.2.4.2.3 of NEN 6771. Working out the effective section properties for slender sections has already been done in STAAD.Pro.

For I-section profiles and tubular sections, the following cases are checked:

1. If $M_{y;s;d} / MN_{y;f;u;d} \leq 1$ check equation 11.2-7 (given below)

$$
V_{z;s;d} / V_{z;u;d} \leq 1
$$

Where

$V_{z;s;d}$ is the shear for in the Z direction

$V_{z;u;d}$ is the shear capacity in the Z direction for ultimate limit state.

For an I section,

$$
V_{z;u;d} = \frac{2}{3} A_{w;et} f_y \cdot \frac{\sqrt{3}}{6}
$$

Where

$A_{w;ef}$ = effective web area as given in section 10.2.4.2.3.

$MN_{y;f;u;d}$ is the moment capacity about the Y axis for the effective section. = ( $f_y \cdot W,_{eff}$)

2. If $M_{y;s;d} / MN_{y;f;u;d} > 1$ and $M_{y;s;d} / M_{y;f;u;d} \leq 1$ check equation 11.2-13 (given below):

$$
\frac{M_{y;s;d}}{M_{N_{y;f;u;d}} + \left( MN_{y;u;d} - M_{N_{y;f;u;d}} \right) \left( 1 - \left( \frac{2V_{z;s;d}}{V_{z;u;d}} - 1 \right) \right)} \leq 1
$$

D5.D.3.5 Clause 6.3 – Buckling resistance of members

The D-NA introduces a new clause 6.3.0, which in turns requires the checks as per clauses 12.1.2.2, 12.13.2 and 12.1.4.2 of NEN 6771 to be applied.

Clause 12.1.2.2 (NEN 6771)

This clause in NEN 6771 determines the relative torsional slenderness and is given as:

$$
\lambda_{\theta,rel} = \sqrt{\frac{N_{cu;ud}}{F_{E;\theta}}}
$$

Where:

$N_{cu;ud} = A f_y \cdot d$
A = area of section
\( f_{y,d} \) = the yield stress
\( F_{E;\theta} \) is the Euler-torsion formula

This value of slenderness is to be used to calculate the modification factors used in section 6.3 of EC-3.

Clause 12.1.3.2 (NEN 6771)

This clause works out the relative torsional-flexural buckling slenderness for compression members. The relative torsional-flexural buckling slenderness is given as:

\[
\lambda_{tk,rel} = \sqrt{\frac{N_{c;u,d}}{F_{E;tk}}}
\]

Where

\( N_{c;u,d} = A \cdot f_{y,d} \)

A = area of section
\( f_{y,d} \) = yield stress
\( F_{E;tk} \) is the Euler torsional buckling strength

Clause 12.1.4.2 (NEN 6771)

Buckling lengths of rotationally restrained bars with intermediate spring supports.

**Note:** STAAD.Pro does not allow for these end conditions, specifically. The effective length factors may be used to accommodate this requirement.

---

**D5.D.3.6 Clause 6.3.1.3 – Slenderness for flexural buckling**

The Dutch NA requires the implementation of clause 12.1.1.3 and 12.1.5.3.2 of NEN 6770 and clause 12.1.1.3 of NEN 6771.

Clause 12.1.1.3 (NEN 6770)

This clause gives the equations to evaluate the effective lengths for various support conditions. STAAD.Pro uses the effective length factor "K" which allows the user to set/modify the effective lengths for a member.

Clause 12.1.5.3.2 (NEN 6770)

This clause gives methods to evaluate the buckling length of lattice sections. We do not deal with latticed section in the current version of STAAD.Pro. In any case the buckling length can be adjusted using the "K" factor.

Clause 12.1.1.3 (NEN 6771)

This clause again deals with working out the effective lengths of prismatic and non-prismatic rods. Again, the "K" factor in the current implementation of STAAD.Pro is adequate to cater for adjusting the effective lengths as necessary.

**D5.D.3.6 Clause 6.3.1.4 – Slenderness for torsional and torsional-flexural buckling**

The D-NA requires the implementation of clauses 12.1.2 and 12.1.3 of NEN 6770

Clause 12.1.2 (NEN 6770): Torsional stability

IPE, HEA, HEB & HEM sections and pipe sections do not need to be checked for torsional instability.
If torsional checks need to be performed, they should be done according to 12.1.2 of NEN 6771.

Clause 12.1.2 (NEN 6771)
This clause gives the condition to check for torsion instability. The condition being:

\[
\frac{N_{c,s;d}}{a_\theta N_{c,u;d}} \leq 1
\]

Where:
- \(N_{c,s;d}\) = the applied axial load
- \(N_{c,u;d}\) = the axial capacity = \(A \cdot f_y\).

\[
a_\theta = \frac{a_{\theta,d}}{f_{u,d}}
\]

Clause 12.1.3 (NEN 6771): Torsional flexural stability
Doubly symmetric sections need not be checked for torsional flexural instability. However, for I sections that have rigid supports that is not along the axis of the section and any other sections will need to be checked as per clause 12.1.3 of NEN 6771.

Clause 12.1.3 (NEN 6771)
This clause gives the condition to check for torsional flexural instability. The condition being:

\[
\frac{N_{c,s;d}}{a_{t,k} N_{c,u;d}} \leq 1
\]

Where:
- \(N_{c,s;d}\) and \(N_{c,u;d}\) as in clause 12.1.2 above.

**D5.D.3.7 Clauses 6.3.2.2 and 6.3.2.3 – Lateral torsional buckling curves**

Clause 6.3.2.2 – Lateral torsional buckling curves - general
The D-NA states that the values for the imperfection factor, \(\alpha_{LT}\), to be used in equation 6.56 of EC-3 are to be obtained from Table 6.3 of EC-3. These are the values used by STAAD.Pro.

Clause 6.3.2.3 – Lateral torsional buckling curves for rolled sections or equivalent welded sections
The D-NA states that:

1. The values for the:
   - Imperfection factor \(\alpha_{LT0} = 0.4\) (used in equation 6.57 of EC-3)
   - \(B = 0.75\) (used in equation 6.57 of EC-3)

   These are the default values used by the program.

2. The buckling curves shall be selected as per Table 6.5.

3. The reduction factor, \(f\), is given by
   \[
   F = 1 - 0.5(1 - kc)[(1 - 2x(\Delta LT -0.8)^2)]
   \]

   \(kc\) is a correction factor for moment distribution determined from Table 6.6. This value can be specified or calculated by the program using the \(KC\) parameter. Refer to **D5.C.6 Design Parameters** (on page 1776)

   The current implementation of STAAD.Pro conservatively uses a value of \(f = 1.0\).
D5.D.3.8 Clause 6.33 – Uniform members in bending and axial compression

The D-NA recommends the use of the method in Annex B of EC-3 to determine the values of kyy, kyz, kzy and kzz to be used in 6.3.3 (EC-3) checks. STAAD.Pro uses the method in Annex B.

Clause 12.3.1.2.3 (NEN 6770): Rotation/bending capacity

The Dutch NA also requires additional checks as per clause 12.3.1.2.3 of NEN 6770.

The checks given in this clause deals with additional checks for columns that form part of a buttressed or non-buttressed framework. The program uses the ESTIFF parameter with two different values to identify the framework type:

Table 149: Framework parameter ESTIFF values for the Dutch NA

<table>
<thead>
<tr>
<th>ESTIFF value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>(default) Column part of a buttressed framework. Selecting this value will internally perform the checks as per section 1 of clause 12.3.1.2.3</td>
</tr>
<tr>
<td>1</td>
<td>Column is not part of a buttressed framework. Selecting this value will internally perform the checks as per section 2 of clause 12.3.1.2.3</td>
</tr>
</tbody>
</table>

These checks are described below:

1. For columns in buttressed frameworks the buckling length is to be taken based on either
   - the system length or
   - the distance between adjacent lateral supports

   The following conditions should also be satisfied:

   If \( \frac{N_{c;S;d}}{N_{pl;d}} < 0.15 \), no additional checks are required

   If \( \frac{N_{c;S;d}}{N_{pl;d}} \geq 0.15 \) and the steel grade is S235 or S275 then
   \[
   \frac{N_{c;S;d}}{N_{pl;d}} + \frac{\lambda_y}{120} \leq 1
   \]

   Where:
   - \( N_{c;S;d} \) is the axial load in the section
   - \( N_{pl;d} \) = Axial capacity of section = \( A \cdot f_{yd} \)
   - \( \lambda_y \) = Slenderness of the section about the major axis (Y-axis)

   If \( \frac{N_{c;S;d}}{N_{pl;d}} \geq 0.15 \) and the steel grade is S355 then
   \[
   \frac{N_{c;S;d}}{N_{pl;d}} + \frac{\lambda_y}{100} \leq 1
   \]

   Where:
   - \( N_{c;S;d} \) = the axial load in the section
   - \( N_{pl;d} \) = Axial capacity of section = \( A \cdot f_{yd} \)
   - \( \lambda_y \) = Slenderness of the section about the major axis (Y-axis)
2. For columns that are not part of buttressed frameworks the following additional checks need to be done:

If $N_{c;sd}/N_{pl;d} < 0.15$, no additional checks are required

If $N_{c;sd}/N_{pl;d} \geq 0.15$ and the steel grade is S235 or S275 then

$$\frac{N_{c;sd}}{N_{pl;d}} + \frac{\lambda_y}{100} \leq 1$$

Where:

- $N_{c;sd}$ = the axial load in the section and
- $N_{pl;d}$ = Axial capacity of section = $Af_{yd}$
- $\lambda_y$ = Slenderness of the section about the major axis (Y-axis)

If $N_{c;sd}/N_{pl;d} \geq 0.15$ and the steel grade is S355 then

$$\frac{N_{c;sd}}{N_{pl;d}} + \frac{\lambda_y}{80} \leq 1$$

D5.D.4 Norwegian National Annex to EC3


The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that require additional clauses from the Norwegian National Annex are:


EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are $\gamma_{M0}$, $\gamma_{M1}$, and $\gamma_{M2}$. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values:

- Resistance of cross-sections - $\gamma_{M0} = 1.05$
- Resistance of members to instability - $\gamma_{M1} = 1.05$
- Resistance of cross sections to tension - $\gamma_{M2} = 1.25$

The design function in STAAD.Pro sets these values as the default values for the Norwegian-NA (NA 3 is specified).

**Note:** You can change these values through the GM0, GM1, & GM2 design parameters. Refer to D5.C.6 Design Parameters (on page 1776)

**Note:** If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

Refer to the basic code (EC3) for a description of these clauses. The sections below refer to the corresponding clauses in the Norwegian -NA.
D5.D.5 UK National Annex to EC3


Note: Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the UK-NA.

The following clauses are not implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling

The UK NA specifies the value of λc0 for I, H channel or box section to be used in equation 6.59 of BS EN 1993-1-1:2005 as 0.4. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the UK National Annex.

Clause 6.3.2.4(2)B – Modification factor “kfl”

The value of the modification factor kfl to be used in equation 6.60 of BS EN 1993-1-1. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the UK National Annex.

The clauses/sections in EN 1993-1-1:2005 that have been dealt with in the UK National Annex (hereafter referred to as the UK-NA) are:


EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γM0, γM1, and γM2. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the UK National Annex:

- Resistance of cross-sections, γM0 = 1.0
- Resistance of members to instability, γM1 = 1.0
- Resistance of cross sections to tension, γM2 = 1.1

The design function in STAAD.Pro sets these values as the default values for the UK-NA (NA 1 is specified).

Note: You can change these values through the GM0, GM1, & GM2 design parameters. Refer to D5.C.6 Design Parameters (on page 1776)

Note: If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

Caution: The GB1 parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005. Hence, any legacy files that use GB1 parameter will indicate an error message and the user will need to substitute GB1 with GM1 in line with EN 1993-1-1:2005.

D5.D.5.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The UK-NA recommends the use of Table 6.3 and 6.4 of BS EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.
The calculation of the LTB reduction factor $\chi_{LT}$, requires the calculation of the “Elastic Critical Buckling Moment”, $M_{cr}$. The UK National Annex does not specify a particular method to calculate $M_{cr}$. Hence the calculation of $M_{cr}$ has been based on the following NCCI documents:

SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling:

This document provides a method to calculate “$M_{cr}$” specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method in the proposed implementation.

The equation to evaluate $M_{cr}$ is given in the NCCI as:

$$M_{cr} = C_1 \frac{n^2EI_s}{(kL)^2} \left[ \frac{k}{(k_w T_s)}^2 \frac{I_w}{I_s} + \frac{(kL)^2GI_t}{n^2EI_s} + (C_2 z_s^2 - C_2 z_s^2) \right]$$

$C_1$ and $C_2$ are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for $C_1$ and $C_2$ for the different cases as given in the tables below:
Figure 3.1 Member with end moments

Table 3.1 Values of $C_1$ for end moment loading (for $k = 1$)

<table>
<thead>
<tr>
<th>$\psi$</th>
<th>$C_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>+1,00</td>
<td>1,00</td>
</tr>
<tr>
<td>+0,75</td>
<td>1,14</td>
</tr>
<tr>
<td>+0,50</td>
<td>1,31</td>
</tr>
<tr>
<td>+0,25</td>
<td>1,52</td>
</tr>
<tr>
<td>0,00</td>
<td>1,77</td>
</tr>
<tr>
<td>-0,25</td>
<td>2,05</td>
</tr>
<tr>
<td>-0,50</td>
<td>2,33</td>
</tr>
<tr>
<td>-0,75</td>
<td>2,57</td>
</tr>
</tbody>
</table>

Table 3.2 Values of factors $C_1$ and $C_2$ for cases with transverse loading (for $k = 1$)

<table>
<thead>
<tr>
<th>Loading and support conditions</th>
<th>Bending moment diagram</th>
<th>$C_1$</th>
<th>$C_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1,121</td>
<td>0,454</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2,678</td>
<td>1,964</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1,348</td>
<td>0,630</td>
</tr>
<tr>
<td></td>
<td>1805</td>
<td>1,683</td>
<td>1,645</td>
</tr>
</tbody>
</table>
This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

The implementation of EC3 in STAAD.Pro accounts for the loading condition and the bending moment diagram through the \( C_{MM} \) parameter. The first two loading conditions mentioned above and its variants can be dealt with by using the existing values of the \( C_{MM} \) parameter (i.e., 1 to 6). Hence the appropriate values from this NCCI will be used for “C1” and ‘C2’ coefficients depending on the value of \( C_{MM} \) specified. The default value of \( C_{MM} \) is 1, which considers the member as a pin ended member with UDL along its span. The user will also have the option to specify specific values for \( C_1 \) and \( C_2 \) using the \( C_1 \) and \( C_2 \) parameters in the design input mode. Refer to D5.C.6 Design Parameters (on page 1776)

However, for cases with end moments and transverse loading, the NCCI provides graphs to evaluate the \( C_1 \) and \( C_2 \) coefficients. It does not however, provide a set of equations for these graphs. However the “end moments and transverse loading” condition cannot be currently specified in the design input. Hence this implementation will introduce two new values for the \( C_{MM} \) parameter viz.

- \( C_{MM} 7: \) Member with varying end moments and uniform loading.
- \( C_{MM} 8: \) Member with varying end moments and central point load.

For these two conditions, the UK National Annex (nor the NCCI) does not provide equations to evaluate \( C_1 \) and \( C_2 \). Hence in STAAD.Pro the user will have to use the new “C1” & ‘C2’ parameters to input the required values for \( C_1 \) & \( C_2 \) to be used in calculating \( M_{cr} \). For values of 7 or 8 for the \( C_{MM} \) parameter, the program will issue a warning if \( C_1 \) and \( C_2 \) have not been specified.

**Note:** If the NA parameter has not been specified, the program obtains the values of \( C_1 \) and \( C_2 \) from Annex F of DD ENV version of 1993-1-1:1992.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment (\( M_{cr} \)) for uniform mono symmetrical sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for “Tee-Sections” will be worked out using the method in this NCCI.

**Note:** Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections. In any case, the actual LTB capacity will still be worked out as per BS 5950-1 as in the current EC3 implementation.

The equation to evaluate \( M_{cr} \) for mono symmetric sections is given as:

\[
M_{cr} = C_1 \frac{n^2 EI_s}{(k_x L)^2} \left[ \left( \frac{k_x}{k_w} \right)^2 \frac{I_w}{I_s} + \frac{(k_x L)^2 G I_T}{n^2 EI_x} + \left( C_2 z_0 - C_3 z_1 \right)^2 - \left( C_2 z_0 - C_3 z_1 \right) \right]
\]

The factors \( C_1 \), \( C_2 \), and \( C_3 \) are dependent on the end conditions and loading criteria. This implementation will consider \( C_1 \), \( C_2 \), and \( C_3 \) as given in the tables below:
The CMM parameter (see section (i) above) specified during design input will determine the values of C1, C2 and C3. The default value of CMM is 1, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the "end moments and transverse loading" condition. The user however can use the new C1, C2 and C3 parameters to input the required values for C1, C2 and C3 to be used in calculating Mcr. As described in section (i) above, the user must use C1, C2 and C3 parameters along with CMM values of 7 and 8.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). The current implementation of EC3 in STAAD takes into account of the end conditions using the CMN parameter. A value of K = kw = 1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints, i.e., CMN = 1.0. For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), the proposed implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) the proposed implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term "zg" in the equation to calculate Mcr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of 'zg' is considered positive if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The user will be allowed to modify this value by using the new 'ZG' parameter. Specifying a value of

---

**Table 4.1** Values of $C_1$, $C_2$, and $C_3$ for end moment loading (for $k_e = 1$)

<table>
<thead>
<tr>
<th>$k_e$</th>
<th>$C_1$</th>
<th>$C_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>0.75</td>
<td>1.14</td>
<td>0.99</td>
</tr>
<tr>
<td>0.50</td>
<td>1.31</td>
<td>0.99</td>
</tr>
<tr>
<td>0.25</td>
<td>1.62</td>
<td>0.99</td>
</tr>
<tr>
<td>0.00</td>
<td>1.77</td>
<td>0.94</td>
</tr>
<tr>
<td>-0.25</td>
<td>2.05</td>
<td>0.85</td>
</tr>
<tr>
<td>-0.50</td>
<td>2.33</td>
<td>0.85</td>
</tr>
<tr>
<td>-0.75</td>
<td>2.57</td>
<td>0.73</td>
</tr>
<tr>
<td>-1.00</td>
<td>2.55</td>
<td>0.60</td>
</tr>
</tbody>
</table>

**Table 4.2** Values effects of $C_1$, $C_2$, and $C_3$ for cases with transverse loading (for $k_e = 1$)

<table>
<thead>
<tr>
<th>Loading and support conditions</th>
<th>Bending moment diagram</th>
<th>$C_1$</th>
<th>$C_2$</th>
<th>$C_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1.13</td>
<td>0.45</td>
<td>0.52</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.57</td>
<td>1.55</td>
<td>0.75</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.36</td>
<td>0.63</td>
<td>1.73</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.65</td>
<td>1.64</td>
<td>2.64</td>
</tr>
</tbody>
</table>
ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term ‘zg’ in the equation will have a value of zero.

D5.D.5.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The UK-NA specifies different values for the $\lambda_{LT,0}$ and $\beta$ factors to be used in equation 6.57 of BS EN 1993-1-1 for rolled and equivalent welded sections. The current implementation in STAAD.pro does not differentiate between rolled and welded sections and uses the default values in BS EN 1993-1-1 for $\lambda_{LT,0}$ and $\beta$. The values specified in the UK-NA are:

- For rolled sections and hot-rolled & cold formed hollow sections:
  $\lambda_{LT,0} = 0.4$
  $\beta = 0.75$
- For welded sections:
  $\lambda_{LT,0} = 0.2$
  $\beta = 1.00$

The current implementation of STAAD.Pro uses the buckling curves based on Table 6.5 of BS EN 1993-1-1:2005. The UK-NA specifies different limits and buckling curves to be used in this clause as given below:

### Table 150: Buckling curves to use with BS EN 1993-1-1:2005

<table>
<thead>
<tr>
<th>Cross Section</th>
<th>Limits</th>
<th>Buckling Curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rolled doubly symmetric I and H sections and hot-finished hollow sections</td>
<td>$h/b \leq 2$</td>
<td>b</td>
</tr>
<tr>
<td></td>
<td>$2.0 &lt; h/b \leq 3.1$</td>
<td>c</td>
</tr>
<tr>
<td></td>
<td>$h/b &gt; 3.1$</td>
<td>d</td>
</tr>
<tr>
<td>Angles (for moments in the major principle plane)</td>
<td>d</td>
<td></td>
</tr>
<tr>
<td>All other hot-rolled sections</td>
<td>d</td>
<td></td>
</tr>
<tr>
<td>Welded, doubly symmetric sections and cold-formed hollow sections</td>
<td>$h/b \leq 2$</td>
<td>c</td>
</tr>
<tr>
<td></td>
<td>$2.0 &lt; h/b \leq 3.1$</td>
<td>d</td>
</tr>
</tbody>
</table>

This table again does not specify which buckling curve is to be used in case of welded doubly symmetric sections with $h/b \geq 3.1$ and welded non-doubly symmetric sections. Hence for these cases the new implementation will still use the method specified in the base code as per clause 6.3.2.2(2).

D5.D.5.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, $\chi_{LT}$ as per UK NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor $\chi_{LT}$ to be used in eqn. 6.55 of BS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating $\chi_{LT}$. Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".
Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of $\chi_{LT}$ should be determined from...”. Hence in the implementation of EC3 (and the UK Annex) in STAAD.Pro, by default the program will consider clause Cl. 6.3.2.3 to evaluate $\chi_{LT}$. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

Cl. 6.3.2.3 in the UK National Annex states that Table 6.5 in BS EN 1993-1-1:2005 should be replaced with the table given in the NA (See section 4.3 of this document). Hence for all cases dealt with by the table in the UK NA, this implementation will choose the buckling curves from the UK National Annex. For any case that is not dealt with by the table in the UK NA, the program will use the method given in Cl. 6.3.2.2 of BS EN 1993-1-1:2005.

Hence for the following cross sections the program will use the Table in the UK NA for choosing a buckling curve for LTB checks (when the UK NA has been specified):

- Rolled doubly symmetric I & H Sections
- Rolled doubly symmetric hollow sections (SHS, RHS, CHS)
- Angle Sections
- Any other rolled section
- Welded doubly symmetric sections with $h/b < 3.1$

For the following cross sections, the program will use Cl. 6.3.2.3 of BS EN 1993-1-1:2005 to evaluate $\chi_{LT}$

- Welded I & H Sections with $h/b \geq 3.1$.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

In any case the Elastic critical moment “Mcr” (used to evaluate the non dimensional slenderness) will be worked out as given in section 4.2 of this document. Since the UK National Annex uses the NCCIs mentioned in the sections above, this implementation will only consider end restraint conditions corresponding to the CMN parameter=1.0 (See section 4.2 above). For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

D5.D.5.5 Clause 6.3.2.3(2) – Modification factor, $f$, for LTB checks

The UK NA specifies the use of eqn. 6.58 of BS EN 1993-1-1:2005 to evaluate the modification factor, $f$, for the LTB reduction factor $\chi_{LT}$. To evaluate the modification factor BS EN 1993-1-1:2005 uses a correction factor $k_c$ given by Table 6.6 in the code.

The UK-NA however, specifies that the correction factor, $k_c$, is to be obtained as below:

$$k_c = 1 / \sqrt{C_1},$$

where $C_1$ is to be obtained from the NCCI documents given in section 4.2 of this document. The NCCI document SN003a-EN-EU specifies the values of $C_1$ to be used in table 3.1 as shown below. This proposed implementation will allow for the reduction factor based on the UK-NA.
Figure 3.1 Member with end moments

Table 3.1 Values of $C_1$ for end moment loading (for $k = 1$)

<table>
<thead>
<tr>
<th>$\psi$</th>
<th>$C_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>+1,00</td>
<td>1,00</td>
</tr>
<tr>
<td>+0,75</td>
<td>1,14</td>
</tr>
<tr>
<td>+0,50</td>
<td>1,31</td>
</tr>
<tr>
<td>+0,25</td>
<td>1,52</td>
</tr>
<tr>
<td>0,00</td>
<td>1,77</td>
</tr>
<tr>
<td>-0,25</td>
<td>2,05</td>
</tr>
<tr>
<td>-0,50</td>
<td>2,33</td>
</tr>
<tr>
<td>-0,75</td>
<td>2,57</td>
</tr>
</tbody>
</table>
These values are for an end restraint factor of k=1 (i.e., CMN=1.0). Hence for all other values of CMN (i.e., 0.7 or 0.5) this implementation will use the values of C1 from DD ENV 1993-1-1:1992 Annex F.

The program will use a default value of 1.0 for kc. However, the user can also input a custom value of kc by setting the design parameter KC to the desired value. The user can also get the program to calculate the value of kc automatically by setting the value of the KC parameter in the design input to 0. This will cause the program to evaluate a value of C1 corresponding to the end conditions and the Bending moment of the member and in turn calculate kc as given in the NA. To evaluate C1, the program will use the NCCI documents mentioned in section 4.2 of this document.

**D5.D.5.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz**

The UK-NA recommends that the method in Annex A or Annex B of BS EN 1993-1-1:2005 can be used to calculate the interaction factors for Cl. 6.3.3 checks in the case of doubly symmetric sections. The proposed implementation will hence use equations in Annex B of BS EN 1993-1-1:2005 to calculate these interaction factors for doubly symmetric sections. The current implementation of EC3 BS in STAAD.pro uses the method in Annex B.

However, for non-doubly symmetric sections, the UK NA gives the option of using Annex B with some modifications as given in the NA. (Cl. NA-3.2 of the UK NA). The UK NA requires additional checks to be done to check for the maximum allowable values of λ and X to be used in equations 6.61 and 6.62 of BS EN 1993-1-1:2005.

As per the UK NA, for non-doubly symmetric sections, the slenderness about the weak axis (λy in STAAD) and the corresponding reduction factor χy should be taken as the values from the highest values of slenderness (λ) among the flexural buckling slenderness (λy), torsional slenderness (λT) and torsional-flexural slenderness (λTF) as given in Clauses 6.3.1.3 and 6.3.1.4 of BS EN 1993-1-1:2005. Hence, for non-doubly symmetric sections, the program will calculate the critical non-dimensional slenderness as:

\[
\lambda_T = \sqrt{\frac{A f_y}{N_{cr}}}
\]

Where:

\[
N_{cr} = \min (N_{crT}, N_{crTF}).
\]
The UK NA or EC3 does not however specify a method to evaluate NcrT or NcrTF. Hence this implementation will use the method specified in the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” to calculate these. See section 4.9 below for details.

**Note:** The UK National Annex or EC3 does not deal with angle sections in specific and hence this implementation will use the method used in the current EC3 implementation to deal with slenderness of angle sections. In the current implementation this is done as per cl 4.7.10 of BS 5950. This proposed implementation will still use the same method for single and double angle sections to evaluate the slenderness.

Clause NA 3.2 of the UK NA also requires that "Where the section is not an I Section or a hollow section and is a class 1 or class 2 section, it will be treated as a class 3 section for the purposes of this clause”. Hence for all Class 1 or Class 2 cross sections that are NOT I, H, SHS, RHS or CHS sections, the elastic properties will be used for the purposes of 6.3.3 checks.

**D5.D.5.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling**

Equations 6.52 and 6.53 of BS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness $\lambda_T$, to be used for torsional and torsional-flexural buckling checks. BS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,TF}$ and $N_{cr,T}$ (refer 6.3.14 of BS EN 1993-1-1:2005).

The NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the $N_{cr,TF}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the UK NA.

The critical axial load for torsional buckling is evaluated as:

$$i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2$$

$i_y$ and $i_z$ = the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2(i_y^2 + i_z^2)} \left[ N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}i_y^2 + i_z^2} \right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

**D5.D.6 French National Annex to EC3**


The following clauses are *not* implemented in STAAD.Pro:

- **Clause 6.3.2.4(1) B – Slenderness for flexural buckling**
  - STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the French National Annex.

- **Clause 6.3.2.4(2)B – Modification factor, kfl**
  - STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the French National Annex.

**Note:** Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the French-NA.
The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the French National Annex (hereafter referred to as FR-NA) and that are relevant to the proposed implementation are:

**D5.D.6.1 Clause 3.2.1(1) - Material Properties**

The material strengths (i.e., steel grade strengths) to be used with NF EN 1993-1-1 are given in Table 3.1 of the code. The French National Annex however, specifies a separate table (Table 3.1 NF) for the yield and tensile strengths of steel grades. This new table replaces Table 3.1 in NF EN 1993-1-1:2005. Table 3.1 NF excludes steel grades from standards EN 10210-1 and EN 10219-1 that are given in EC-3.

STAAD.Pro uses the steel grades and values from the table given in the National Annex (i.e., Table 3.1 NF). Table 3.1 NF is similar to table 3.1 in EC3, apart from the $f_u$ values for S 355 and S355 W grade steel.

**Table 151: Material strengths specified for use with the NF-NA**

<table>
<thead>
<tr>
<th>Standard and grade of steel</th>
<th>Nominal thickness, $t$, of the element (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$t &lt; 40$ mm</td>
</tr>
<tr>
<td></td>
<td>$f_y$ (N/mm$^2$)</td>
</tr>
<tr>
<td>EN 10025-2</td>
<td></td>
</tr>
<tr>
<td>S 235</td>
<td>235</td>
</tr>
<tr>
<td>S 275</td>
<td>275</td>
</tr>
<tr>
<td>S 355</td>
<td>355</td>
</tr>
<tr>
<td>S 450</td>
<td>440</td>
</tr>
<tr>
<td>EN 10025-3</td>
<td></td>
</tr>
<tr>
<td>S 275 N/NL</td>
<td>275</td>
</tr>
<tr>
<td>S 355 N/NL</td>
<td>355</td>
</tr>
<tr>
<td>S 420 N/NL</td>
<td>420</td>
</tr>
<tr>
<td>S 460 N/NL</td>
<td>460</td>
</tr>
<tr>
<td>EN 10025-4</td>
<td></td>
</tr>
<tr>
<td>S 275 M/ML</td>
<td>275</td>
</tr>
<tr>
<td>S 355 M/ML</td>
<td>355</td>
</tr>
<tr>
<td>S 420 M/ML</td>
<td>420</td>
</tr>
<tr>
<td>S 460 M/ML</td>
<td>460</td>
</tr>
<tr>
<td>EN 10025-5</td>
<td></td>
</tr>
<tr>
<td>S 235 W</td>
<td>235</td>
</tr>
<tr>
<td>S 355 W</td>
<td>355</td>
</tr>
<tr>
<td>EN 10025-6</td>
<td></td>
</tr>
<tr>
<td>S 460 Q/QL/QL 1</td>
<td>460</td>
</tr>
</tbody>
</table>
If you specify a steel grade that is not given in the Annex Table 3.1 (NF) but is present in Table 3.1 of EN 1993-1-1:2005, the program uses the values from Table 3.1 of EN 1993-1-1:2005. The appropriate yield strength (f_y) used is shown in the design output file.

**D5.D.6.2 Clause 6.1(1) – General**

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_M0, γ_M1, and γ_M2. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the French National Annex:

- Resistance of cross-sections, γ_M0 = 1.0
- Resistance of members to instability, γ_M1 = 1.0
- Resistance of cross sections to tension, γ_M2 = 1.25

The design function in STAAD.Pro sets these values as the default values for the NF-NA (NA 4 is specified).

**Note:** You can change these values through the GM0, GM1, & GM2 design parameters. Refer to [D5.C.6 Design Parameters](#) (on page 1776)

**Note:** If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

**Caution:** The GB1 parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005. Hence, any legacy files that use GB1 parameter will indicate an error message and the user will need to substitute GB1 with GM1 in line with EN 1993-1-1:2005.

**D5.D.6.3 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks**

The French NA recommends the use of Table 6.3 and 6.4 of NF EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_LT requires the calculation of the “Elastic Critical Buckling Moment”, Mcr. The French NA gives a method to evaluate Mcr in its “Annex MCR”. This implementation will make use of this method to evaluate Mcr. Annex MCR however deals with the calculation of Mcr for doubly symmetric sections. Hence this implementation will use this method only for doubly symmetric sections. For mono symmetric sections that are symmetric about the minor axis (i.e Tee sections) this implementation will use the method from the NCCI document SN030a-EN-EU as given in the section below. For any other type of section that is not dealt with by the Annex, this implementation will use the method and tables given in Annex F of DD ENV 1993-1-1:1992.

**Annex MCR**

This document provides a method to calculate Mcr specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method in this implementation.

The equation to evaluate Mcr is given as:

\[
M_{cr} = C_1 \frac{n^2 E I_s}{(kL)^2} \left[ \left( \frac{k}{k_w} \right)^2 \frac{I_w}{T_s} + \frac{(kL)^2 G I_t}{n^2 E I_s} \right] + \left( C_2 z_s \right)^2 - \left( C_2 z_s \right) 
\]
C_1 and C_2 are factors that depend on the end conditions and the loading conditions. The NCCI provides values for C_1 and C_2 for the different cases as given in Table 1 and Table 2 of the Annex. Table 1 deals with the condition of a simply supported member with end moments and the value of C_1 is determined by the end moment ratio (Refer to the NA for details). Clause 3.2 of the National Annex however gives a formula to evaluate C_1 as:

\[
C_1 = \frac{1}{\sqrt{0.325 + 0.423\psi + 0.252\psi^2}}
\]

This formula however does not match the values given in Table 1 of the NA. Hence this implementation will use the values of C_1 from Table 1 if the end moment ratio (\(\psi\)) is exactly equal to the values of \(\psi\) in the table. For all other cases this implementation will calculate the value of C_1 from equation (6) in the Annex.

The value of C_2 will be determined from Table 2 of the Annex based on the loading and end conditions (i.e. the CMM parameter in STAAD).

The user will also have the option to specify specific values for C_1 and C_2 using the C1 and C2 parameters in the design input mode. Refer to D5.C.6 Design Parameters (on page 1776)

The French NA considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

The first two cases and its variants can be defined using with the existing CMM parameter values in STAAD.Pro. However the third condition cannot be currently specified in the design input. Hence this implementation will introduce two new values for CMM viz.

CMM 7: Member with varying end moments and uniform loading.

CMM 8: Member with varying end moments and central point load.

The load to moment ratio (\(\mu\)) will then be used in the calculations will then be used to calculate C1 and C2 as given in section 3.5 of Annex MCR (See Annex MCR in the NA for details).

This implementation will also introduce a new parameter “MU” to be specified when using CMM = 7 or 8. The load to moment ratio (\(\mu\)) to be used in the calculations is to be input using the new ‘MU’ parameter. This implementation will require that for the French National Annex if CMM = 7 or 8 has been specified, the user should also either specify a value for ‘MU’ or input the values for C1 and C2 using the ‘C1’ and/or ‘C2’ parameters directly.

**Note:** The new parameter MU will currently be applicable only in the context of the French NA.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment (Mcr) for uniform mono-symmetric sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for “Tee-Sections” will be worked out using the method in this NCCI.

**Note:** Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate M_{cr} for mono-symmetric sections is given as:

\[
M_{cr} = C_1 \left[ \left( \frac{k_x}{k_w} \right)^2 I_w + \frac{(k_x L)^2 G I_T}{n^2 E I_x} + \left( C_2 z_e - C_3 z_1 \right)^2 - \left( C_2 z_e - C_3 z_1 \right)^2 \right]
\]
The factors $C_1$, $C_2$, and $C_3$ are dependent on the end conditions and loading criteria. This implementation will consider $C_1$, $C_2$, and $C_3$ as given in the tables below:

<table>
<thead>
<tr>
<th>$\psi$</th>
<th>$C_1$</th>
<th>$C_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>+1.00</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>+0.75</td>
<td>1.14</td>
<td>0.99</td>
</tr>
<tr>
<td>+0.50</td>
<td>1.31</td>
<td>0.99</td>
</tr>
<tr>
<td>+0.25</td>
<td>1.52</td>
<td>0.99</td>
</tr>
<tr>
<td>0.00</td>
<td>1.71</td>
<td>0.94</td>
</tr>
<tr>
<td>-0.25</td>
<td>2.05</td>
<td>0.86</td>
</tr>
<tr>
<td>-0.50</td>
<td>2.33</td>
<td>0.88</td>
</tr>
<tr>
<td>-0.75</td>
<td>2.57</td>
<td>0.81</td>
</tr>
<tr>
<td>-1.00</td>
<td>2.55</td>
<td>0.80</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Loading and support conditions</th>
<th>Bending moment diagram</th>
<th>$C_1$</th>
<th>$C_2$</th>
<th>$C_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1.13</td>
<td>0.46</td>
<td>0.52</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.57</td>
<td>1.56</td>
<td>0.75</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.36</td>
<td>0.63</td>
<td>1.73</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.80</td>
<td>1.84</td>
<td>2.64</td>
</tr>
</tbody>
</table>

The CMM parameter specified during design input will determine the values of $C_1$, $C_2$ and $C_3$. The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the “end moments and transverse loading” condition. The user however can use the new “$C_1$, ‘$C_2’ and ‘$C_3’ parameters to input the required values for $C_1$, $C_2$ and $C_3$ to be used in calculating Mcr.

**Note:** If “MU” as well as $C_1$, $C_2$ and $C_3$ have been specified, the program will ignore MU and use the user input values of $C_1$, $C_2$ and $C_3$. The current implementation of EC3 in STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Also, the NCCI document and Annex MCR of the FR-NA assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw=1, i.e., CMN parameter =1.0). Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints. For members with partial or end fixities (i.e, CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term “$z_g$” in the equation to calculate Mcr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of ‘$z_g$’ is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the...
shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term “zg” in the equation will have a value of zero.

**Note:** There is a separate method specified in the NCCI document “SN006a-EN-EU” to calculate M\(_{cr}\) for cantilever beams. Again this document does not give any specific formulae to evaluate the coefficients. Hence, this has not been implemented in STAAD.Pro.

### D5.D.6.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The FR-NA provides equations to evaluate the \(\lambda_{LT,0}\) and \(\alpha_{LT}\) factors given in clause 6.3.2.3

For rolled doubly symmetric sections use:

\[
\lambda_{LT,0} = 0.2 + 0.1 \frac{b}{h}
\]
\[
\alpha_{LT} = 0.4 - 0.2 \frac{b}{h} \lambda_{LT}^2 \geq 0
\]

**Note:** Since EN 1993-1-1:2005 limits the value of \(\lambda_{LT,0}\) to 0.4, STAAD.Pro limits \(\lambda_{LT,0}\) to a maximum value of 0.4.

For welded doubly symmetric sections use:

\[
\lambda_{LT,0} = 0.3 \frac{b}{h}
\]
\[
\alpha_{LT} = 0.5 - 0.25 \frac{b}{h} \lambda_{LT}^2 \geq 0
\]

For other sections:

\[
\lambda_{LT,0} = 0.2
\]
\[
\alpha_{LT} = 0.76
\]

And for all sections, \(\beta = 1.0\)

These equations and factors are then applied to equation 6.57 of NF EN 1993-1-1 to evaluate the Lateral Torsional Buckling reduction factor \(\chi_{LT}\).

### D5.D.6.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The French NA specifies that the modification factor is to be obtained as per the default method given in EC-3. Hence this implementation will use the existing functionality to evaluate the correction factor \(k_c\) to be used in the modification factor \(f\).

The program uses a default value of 1.0 for \(k_c\). However the user can also input a custom value of \(k_c\) by setting the design parameter \(K_C\) to the desired value. You may instruct the program to calculate the value of \(k_c\) automatically by setting the value of the \(K_C\) parameter in the design input to 0. This will cause the program to evaluate \(k_c\) from Table 6.6 of NF EN 1993-1-1:2005. This will correspond to the end conditions and the bending moment of the member (i.e., the value of CMM parameter specified).

For CMM = 7, the program will choose the value of \(k_c\) to be either 0.90 or 0.91 based on the end moment ratio.

For CMM = 8, the program will choose the value of \(k_c\) to be either 0.77 or 0.82 based on the end moment ratio.

An additional check will also be performed as given below:
\[ \chi_{LT,\text{mod}} \leq \frac{1}{\chi_{LT}} \]

The French Annex specifies that the modification factor is applicable only to members that are free to rotate on plan (i.e., CMN \(1, \theta\)). Hence for all other values of CMN, this implementation will ignore "f" and hence will use \(\chi_{LT,\text{mod}} = \chi_{LT}\).

**D5.D.6.6 Clause 6.3.3(5) – Interaction factors \(k_{yy}, k_{yz}, k_{zy}, \text{and } k_{zz}\)**

The French NA recommends the use of equations in Annex A of NF EN 1993-1-1:2005 to calculate these interaction factors. STAAD.pro uses the method in Annex B for design per EC3 (without National Annex). Therefore, the method in Annex A has been added into the program.

**Note:** The NA mentions that this method can be extended to singly symmetric I-Sections (symmetric about the minor axis) if the elastic properties are used instead of the plastic properties. However, since STAAD does not have a provision to specify such sections, this case will not be considered for this implementation.

The NA also mentions that torsional flexural buckling needs to be taken into account in case of mono symmetric sections. This is taken into account based on the method given in the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes”. Refer to **D5.D.6.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling** (on page 1819)

The NA also recommends a lower limit as given below for the term \(C_{mi,0}\) in Table A.2 of Annex A:

\[ C_{mi,0} \geq 1 - \frac{N_{Ed}}{N_{cr,i}} \]

**D5.D.6.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling**

Equations 6.52 and 6.53 of NF EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness \(\lambda_T\), to be used for torsional and torsional-flexural buckling checks. NF EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads \(N_{cr,T,F}\) and \(N_{cr,T}\) (refer 6.3.14 of NF EN 1993-1-1:2005).

The NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the \(N_{cr,T,F}\) and \(N_{cr,T}\) factors and therefore these methods are used to evaluate the elastic critical loads for the French NA.

The critical axial load for torsional buckling is evaluated as:

where

\[ i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2 \]

\[ i_y \text{ and } i_z \text{ are the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively} \]

The critical axial load for torsional-flexural buckling is evaluated as:

\[ N_{cr,T,F} = \frac{i_o^2}{2(i_y^2 + i_z^2)} \left[ N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T} \frac{i_y^2 + i_z^2}{i_o^2}} \right] \]

For details on these equations, refer to the NCCI document SN001a-EN-EU.
D5.D.7 Finnish National Annex to EC3


The following clauses are not implemented in STAAD.Pro:

**Clause 6.3.2.4(1) B – Slenderness for flexural buckling**

STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Finnish National Annex.

**Clause 6.3.2.4(2)B – Modification factor “kfl”**

STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Finnish National Annex.

*Note:* Refer to the [basic code (EC3)](on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the Finnish-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Finnish National Annex (hereafter referred to as SFS-NA) and that are relevant to the proposed implementation are:

**D5.D.7.1 Clause 3.2.1(1) - Material Properties**

The material strengths (i.e., steel grade strengths) to be used with SFS-EN 1993-1-1 are given in Table 3.1 of the code. These steel grade values are specified using the SGR parameter (Refer to [D5.C.6 Design Parameters](on page 1776)).

The Finnish National Annex states in Cl. 3.1(2) that, apart from the steel grades specified in Table 3.1 of SFS EN 1993-1-1, the following steel grades can also be used:

- Steel grades S315MC, S355MC, S420MC and S460MC according to SFS-EN 10149-2
- Steel grades S260NC, S315NC, S355NC and S420NC according to SFS-EN 10149-3

These grades of steel can be specified by using the PY (Yield Strength) and FU (Ultimate Strength) parameters in STAAD.Pro. Set these parameters to the respective values as given in SFS-EN 10149-2/3 for the steel grades specified above. The choice of the buckling curve to be used is based on the value of the SGR parameter specified. The output will include the appropriate yield strength used for design.

**D5.D.7.2 Clause 6.1(1) – General**

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are $\gamma_{M0}$, $\gamma_{M1}$, and $\gamma_{M2}$. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Finnish National Annex:

- Resistance of cross-sections, $\gamma_{M0} = 1.0$
- Resistance of members to instability, $\gamma_{M1} = 1.0$
- Resistance of cross sections to tension, $\gamma_{M2} = 1.25$

The design function in STAAD.Pro sets these values as the default values for the SFS-NA (NA 5 is specified).

*Note:* You can change these values through the GM0, GM1, & GM2 design parameters. Refer to [D5.C.6 Design Parameters](on page 1776)
Note: If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

Caution: The GB1 parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005. Hence, any legacy files that use GB1 parameter will indicate an error message and the user will need to substitute GB1 with GM1 in line with EN 1993-1-1:2005.

D5.D.7.3 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks

The Finnish NA recommends the use of Table 6.3 and 6.4 of SFS EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor \( \chi_{LT} \), requires the calculation of the “Elastic Critical Buckling Moment”, \( M_{cr} \). The Finnish National Annex does not specify a particular method to calculate \( M_{cr} \). Hence the calculation of \( M_{cr} \) has been based on the following NCCI documents:

SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling:

This document provides a method to calculate \( M_{cr} \) specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method. The equation to evaluate \( M_{cr} \) is given in the NCCI as:

\[
M_{cr} = C_1 \frac{\pi^2 E I_s}{(kL)^2} \left[ \left( \frac{k}{k_w} \right)^2 \frac{I_w}{I_s} \left( \frac{kL}{n^2 E I_s} \right)^2 \left( C_2 \frac{z_s}{I_w} \right)^2 + \left( C_2 \frac{z_s}{I_s} \right)^2 - C_2 \right]
\]

\( C_1 \) and \( C_2 \) are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for \( C_1 \) and \( C_2 \) for the different cases as given in the tables below:

**Table 152: Values of \( C_1 \) for end moment loading (for k=1)**

<table>
<thead>
<tr>
<th>( \psi )</th>
<th>( C_1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>+1,00</td>
<td>1,00</td>
</tr>
<tr>
<td>+0,75</td>
<td>1,14</td>
</tr>
<tr>
<td>+0,50</td>
<td>1,31</td>
</tr>
<tr>
<td>+0,25</td>
<td>1,52</td>
</tr>
<tr>
<td>0,00</td>
<td>1,77</td>
</tr>
<tr>
<td>-0,25</td>
<td>2,05</td>
</tr>
<tr>
<td>-0,50</td>
<td>2,33</td>
</tr>
</tbody>
</table>
This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the CMM parameter.

\[
\psi \quad C_1
\]

<table>
<thead>
<tr>
<th>(\psi)</th>
<th>(C_1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-0.75</td>
<td>2.57</td>
</tr>
</tbody>
</table>

This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the CMM parameter.

SN030a-EN-EU - Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment \(M_{cr}\) for uniform mono symmetric sections that are symmetric about the weak axis. Hence, the elastic critical moment for “Tee-Sections” will be worked out using the method in this NCCI.

**Note:** Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate \(M_{cr}\) for mono symmetric sections is given as:

\[
M_{cr} = C_1 \frac{n^2 EI_s}{(k_x L)^2} \left[ \left( \frac{k_x}{k_w} \right)^2 \frac{I_w}{I_s} + \frac{(k_x L)^2 G I_T}{n^2 EI_x} + (C_2 z_e - C_3 z_1)^2 - \left( C_2 z_e - C_3 z_1 \right) \right]
\]

The factors \(C_1\), \(C_2\), and \(C_3\) are dependent on the end conditions and loading criteria. This implementation will consider \(C_1\), \(C_2\), and \(C_3\) as given in the tables below:
The CMM parameter specified during design input will determine the values of C₁, C₂, and C₃. The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the “end moments and transverse loading” condition. You can use the C₁, C₂, and C₃ parameters to input the required values for C₁, C₂, and C₃ to be used in calculating M_cr.

Note: If MU as well as C₁, C₂, and C₃ have been specified, the program will ignore M.U and use the user input values of C₁, C₂, and C₃. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term “zg” in the equation to calculate M_cr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of ‘zg’ is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0
in the design input would indicate that the load acts exactly at the shear center of the section so that the term “zg” in the equation will have a value of zero.

**Note:** The program does not consider the case of cantilevers.

### D5.D.7.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The Finnish-NA provides the values for the terms $\lambda_{LT,0}$ and $\beta$ factors given in clause 6.3.2.3(1) as follows:

For rolled doubly symmetric sections and hollow sections, use:

$$\lambda_{LT,0} = 0.4 \text{ and } \beta = 0.75$$

For welded doubly symmetric sections and hollow sections use:

$$\lambda_{LT,0} = 0.2 \text{ and } \beta = 1.0$$

The Finnish NA specifies the following limits for choosing the buckling curves:

**Table 153: Selection of lateral torsional buckling curve for cross sections using equation (6.57)**

<table>
<thead>
<tr>
<th>Cross-section (constant cross-section)</th>
<th>Limits</th>
<th>Buckling Curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rolled double symmetric I- and H-sections and hot finished hollow sections.</td>
<td>$h/b \leq 2$&lt;br&gt;$2 &lt; h/b &lt; 3.1$</td>
<td>b&lt;br&gt;c</td>
</tr>
<tr>
<td>Welded double symmetric I-section and H-sections and cold-formed hollow sections</td>
<td>$h/b \leq 2$&lt;br&gt;$2 &lt; h/b &lt; 3.1$</td>
<td>c&lt;br&gt;d</td>
</tr>
</tbody>
</table>

The NA says that for all other cases the rules given in Cl 6.3.2.2 should be used. Hence even for rolled or welded doubly symmetric sections with $h/b$ ratio $\geq 3.1$, this implementation will resort to checks as per clause 6.3.2.2.

These equations and factors are then applied to equation 6.57 of SFS-EN 1993-1-1 to evaluate the Lateral Torsional Buckling reduction factor $\chi_{LT}$.

### D5.D.7.5 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, $\chi_{LT}$ as per Finnish NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor $\chi_{LT}$ to be used in eqn. 6.55 of SFS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating $\chi_{LT}$. Table 6.4 specifies the choice of buckling curves for “Rolled I Sections”, “Welded I Sections” and “Any other sections”. Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of $\chi_{LT}$ should be determined from...”. Hence in the implementation of EC3 (and the Finnish Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate $\chi_{LT}$. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate $\chi_{LT}$. 

---

**Design**

**D. Design Codes**

**STAAD.Pro** 1824 **User Manual**
Cl. 6.3.2.3 in the Finnish National Annex gives equations to evaluate the imperfection factors to be used for various section types (Refer to D5.D.7.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1821)). Hence for all cases dealt with by the equations in the Finnish NA, this implementation will use Cl 6.3.2.3 to evaluate $\chi_{LT}$.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

In any case, the elastic critical moment, $M_{cr}$, (used to evaluate the non dimensional slenderness) will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the $CMN$ parameter=1.0 (Refer to D5.D.7.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1821)) will be considered. For all other cases of the $CMN$ parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

**Note:** If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

**D5.D.7.6 Clause 6.3.2.3(2) – Modification factor, $f$, for LTB checks**

STAAD.Pro uses the value of the modification factor $f = 1.0$ as given in the Finnish NA.

**D5.D.7.7 Clause 6.3.3(5) – Interaction factors $k_{yy}$, $k_{yz}$, $k_{zy}$, and $k_{zz}$**

The Finnish NA recommends the use of equations in Annex A or Annex B of SFS-EN 1993-1-1 to calculate these interaction factors. STAAD.Pro uses the method in Annex B by default. This implementation of the Finnish NA will also use Annex B for Cl.6.3.3 checks.

**D5.D.7.8 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling**

Equations 6.52 and 6.53 of SFS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness $\lambda_T$, to be used for torsional and torsional-flexural buckling checks. SFS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of SFS EN 1993-1-1:2005).

The NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the $N_{cr,T,F}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the Finnish NA.

The critical axial load for torsional buckling is evaluated as:

$$N_{cr,T} = \frac{i_o^2}{i_y^2 + i_z^2} \left( N_{cr,y} + N_{cr,T} \right) - \sqrt{\left( N_{cr,y} + N_{cr,T} \right)^2 - 4 N_{cr,y} N_{cr,T} \left( i_y^2 + i_z^2 \right)}$$

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,T,F} = \frac{i_o^2}{2(i_y^2 + i_z^2)} \left( N_{cr,y} + N_{cr,T,F} \right) - \sqrt{\left( N_{cr,y} + N_{cr,T,F} \right)^2 - 4 N_{cr,y} N_{cr,T,F} \left( i_y^2 + i_z^2 \right)}$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.
D5.D.8 Polish National Annex to EC3


The following clauses are not implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling

STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Polish National Annex.

Clause 6.3.2.4(2)B – Modification factor “kfl”

STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Polish National Annex.

Note: Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the Polish-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Polish National Annex, NA for PN-EN 1993-1-1:2006, (hereafter referred to as PN-NA) and that are relevant to the proposed implementation are:

D5.D.8.1 Clause 3.2.1(1) - Material Properties

The material strengths (i.e., steel grade strengths) to be used with PN-EN 1993-1-1 are given in Table 3.1 of the code. The Polish National Annex states in Cl. 3.1(2) that the steel grades to be used will be based on Table 3.1 of PN EN 1993-1-1. These steel grade values are specified using the SGR parameter (Refer to D5.C.6 Design Parameters (on page 1776)).

D5.D.8.2 Clause 6.1(1) – General

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in design as given in Cl. 6.1 of the code. These factors are \( \gamma_{M0} \), \( \gamma_{M1} \), and \( \gamma_{M2} \). EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Polish National Annex:

- Resistance of cross-sections, \( \gamma_{M0} = 1.0 \)
- Resistance of members to instability, \( \gamma_{M1} = 1.0 \)
- Resistance of cross sections to tension, \( \gamma_{M2} = \) minimum of 1.1 or \( 0.9 \times \frac{f_u}{f_y} \)

Where:

\[ f_u \] is the ultimate steel strength

\[ f_y \] is the yield strength of steel

The design function in STAAD.Pro sets these values as the default values for the PN-NA (NA 6 is specified).

Note: You can change these values through the GM0, GM1, & GM2 design parameters. Refer to D5.C.6 Design Parameters (on page 1776)

Note: If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

Caution: The GB1 parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005.
Hence, any legacy files that use GB1 parameter will indicate an error message and the user will need to substitute GB1 with GM1 in line with EN 1993-1-1:2005.

**D5.D.8.3 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks**

The Polish NA recommends the use of Table 6.3 and 6.4 of PN EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor $\chi_{LT}$ requires the calculation of the “Elastic Critical Buckling Moment”, $M_{cr}$. The Polish National Annex does not specify a particular method to calculate $M_{cr}$. Hence the calculation of $M_{cr}$ has been based on the following NCCI documents:

**SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling:**

This document provides a method to calculate $M_{cr}$ specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method. The equation to evaluate $M_{cr}$ is given in the NCCI as:

$$M_{cr} = C_1 \frac{n^2 EI}{(kL)^2} \left[ \frac{2 E I_w}{I_s} + \frac{(kL)^2 G I_t}{n^2 EI_s} + (C_2 z_s)^2 - C_2 z_s \right]$$

$C_1$ and $C_2$ are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for $C_1$ and $C_2$ for the different cases as given in the tables below:

**Table 154: Values of $C_1$ for end moment loading (for k=1)**

<table>
<thead>
<tr>
<th>$\psi$</th>
<th>$C_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>+1,00</td>
<td>1,00</td>
</tr>
<tr>
<td>+0,75</td>
<td>1,14</td>
</tr>
<tr>
<td>+0,50</td>
<td>1,31</td>
</tr>
<tr>
<td>+0,25</td>
<td>1,52</td>
</tr>
<tr>
<td>0,00</td>
<td>1,77</td>
</tr>
<tr>
<td>-0,25</td>
<td>2,05</td>
</tr>
<tr>
<td>-0,50</td>
<td>2,33</td>
</tr>
<tr>
<td>-0,75</td>
<td>2,57</td>
</tr>
</tbody>
</table>

This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the CMM parameter.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment \(M_{cr}\) for uniform mono-symmetric sections that are symmetric about the weak axis. Hence, the elastic critical moment for “Tee-Sections” will be worked out using the method in this NCCI.

**Note:** Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate \(M_{cr}\) for mono-symmetric sections is given as:

\[
M_{cr} = C_1 \left( \frac{n^2 EI_s}{(k_x L)^2} \right)^2 \left( \frac{k_x}{T_w} \right)^2 + \left( \frac{(k_x L)^2 G I_T}{n^2 EI_s} \right)^2 + (C_2 z_c - C_3 z_1)^2 - \left( C_2 z_c - C_3 z_1 \right)
\]

The factors \(C_1, C_2, \text{ and } C_3\) are dependent on the end conditions and loading criteria. This implementation will consider \(C_1, C_2, \text{ and } C_3\) as given in the tables below:

<table>
<thead>
<tr>
<th>(\psi)</th>
<th>(C_1)</th>
<th>(C_2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>+0.10</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>+0.15</td>
<td>1.14</td>
<td>0.99</td>
</tr>
<tr>
<td>+0.20</td>
<td>1.31</td>
<td>0.99</td>
</tr>
<tr>
<td>+0.25</td>
<td>1.52</td>
<td>0.96</td>
</tr>
<tr>
<td>+0.30</td>
<td>1.74</td>
<td>0.94</td>
</tr>
<tr>
<td>+0.35</td>
<td>2.05</td>
<td>0.85</td>
</tr>
<tr>
<td>+0.40</td>
<td>2.33</td>
<td>0.85</td>
</tr>
<tr>
<td>+0.50</td>
<td>2.57</td>
<td>0.85</td>
</tr>
<tr>
<td>+0.75</td>
<td>2.55</td>
<td>0.83</td>
</tr>
</tbody>
</table>

The CMM parameter specified during design input will determine the values of \(C_1, C_2, \text{ and } C_3\). The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not
however consider the "end moments and transverse loading" condition. You can use the C1, C2, and C3 parameters to input the required values for C1, C2, and C3 to be used in calculating Mcr.

**Note:** If MU as well as C1, C2, and C3 have been specified, the program will ignore MU and use the user input values of C1, C2, and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term “zg” in the equation to calculate Mcr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of ‘zg’ is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term “zg” in the equation will have a value of zero.

**Note:** The program does not consider the case of cantilevers.

**D5.D.8.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section**

The Polish-NA provides the values for the terms λLT,0 and β factors given in clause 6.3.2.3(1) as follows:

For all sections, use:

\[ \lambda_{LT,0} = 0.4 \text{ and } \beta = 0.75 \]

The Polish NA specifies the use of uses table 6.5 to work out the buckling curves for use in Cl. 6.3.2.3. Hence table 6.5 in PN-EN 1993-1-1 will be used for this.

These equations and factors are then applied to equation 6.57 of PN-EN 1993-1-1 to evaluate the Lateral Torsional Buckling reduction factor \( \chi_{LT} \).

**D5.D.8.5 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, \( \chi_{LT} \) as per Finnish NA**

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor \( \chi_{LT} \) to be used in eqn. 6.55 of PN EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating \( \chi_{LT} \). Table 6.4 specifies the choice of buckling curves for “Rolled I Sections”, “Welded I Sections” and “Any other sections”. Cl 6.3.2.2 on the other hand uses tables 6.5 and 6.6 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of \( \chi_{LT} \) should be determined from...”. Hence in the implementation of EC3 (and the Finnish Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate \( \chi_{LT} \). For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate \( \chi_{LT} \).
Cl. 6.3.2.3 in the Finnish National Annex gives equations to evaluate the imperfection factors to be used for various section types (Refer to D5.D.7.3 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks (on page 1821) ). Hence for all cases dealt with by the equations in the Finnish NA, this implementation will use Cl 6.3.2.3 to evaluate $\chi_{LT}$.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

In any case, the elastic critical moment, $M_{cr}$, (used to evaluate the non-dimensional slenderness) will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the $CMN$ parameter=1.0 (Refer to D5.D.7.3 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks (on page 1821) ) will be considered. For all other cases of the $CMN$ parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1-1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1-2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., $SBLT$ parameter = 1.0 in design input).

\textbf{D5.D.8.6 Clause 6.3.2.3(2) – Modification factor, }$f$, \textbf{for LTB checks}

STAAD.Pro uses the value of the modification factor $f$ as per eqn 6.58 of PN-EN 1993-1-1. The correction factor, $k_c$, will be evaluated as:

$$k_c = \sqrt{(C_{mLT})}$$

Where:

$C_{mLT}$ is the equivalent uniform moment factor from table B.3 of PN-EN 1993-1-1. $C_{mLT}$ is evaluated based on the end conditions of the member and the shape of the bending moment diagram. However, if the $KC$ parameter has been used, then the program will use the specified value.

\textbf{D5.D.8.7 Clause 6.3.3(5) – Interaction factors }$k_{yy}$, $k_{yz}$, $k_{zy}$, \textbf{and }$k_{zz}$

The Polish NA recommends the equations in Annex B of PN-EN 1993-1-1 to calculate these interaction factors. The current implementation of EC3 BS in STAAD.pro uses the method in Annex B by default. The proposed implementation of the Polish NA will also use Annex B for Cl.6.3.3 checks.

The Polish NA also gives two additional simplified checks. This implementation will provide for these additional checks as well. However as they are intended as optional checks, by default, the program will not perform these checks. However, the user can invoke these checks by using the PLG parameter. Refer to \textbf{D5.C.6 Design Parameters} (on page 1776)

If the value of the PLG parameter is set to 1, the following two checks will be performed as per Cl. NA.20.(2) and NA.20(3) respectively:

- \textbf{Cl. NA.20.(2):} The following condition will be checked

\[ \frac{n}{A} + C_{my} \frac{m_y}{\chi_{LT}} + C m_z m \leq 1 - \Delta_0 \]  $(I = y$ or $z)$

where

\[ n = \frac{N_{Ed}}{N_{Rd}} \]

\[ m_y = \max M_{y,Ed} (+ \Delta M_{y,Ed}) / M_{Rd} \]

\[ m_z = \max M_{Z,Ed} (+ \Delta M_{Z,Ed}) / M_{Z} \]

\[ \chi \text{ and} \]

\[ C \text{ buckling factor} \]
\(\chi_{LT}\) = LTB factor
\(C_m\) = moment factor from table B 3 of PN EN 1993-1-1
\(\Delta_0\) = correction factor (estimation of maximum reduction) and will be worked out as:

\[\Delta_0 = 0.1 + 0.2 (w_i - 1),\text{ przy czym } w_i = W_{pl,i}/W_{el,i},\text{ or }\]
\[\Delta_0 = 0.1 - \text{ in case of class 3 and 4 sections}.\]

- Cl. NA.20.(3): This condition will only be checked for circular hollow sections.

\[n/\chi_i + [(k_{ii} m_i)^2 + (C_{mj} m_j)^2]^{1/2} \leq 1 \text{ (i,j =y,z)}\]

where

- \(k\) = the interaction factor from table B.1 of PN-EN 1993-1-1
- \(n, m, \text{ and } C_{mj}\) = as previously described

If the PLG parameter has been set to 1, the maximum among the following ratios will be taken as being critical for Cl 6.3.3:

- 6.3.3: Eqn6.61
- 6.3.3: Eqn6.62
- NA.20(2) and
- NA.20(3)

If however PLG has been set to 0 (or not specified at all), the program will ignore the last two checks in the list above.

**D5.D.8.8 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling**

Equations 6.52 and 6.53 of PN EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness \(\lambda_T\), to be used for torsional and torsional-flexural buckling checks. PN EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads \(N_{cr,T,F}\) and \(N_{cr,T}\) (refer 6.3.14 of PN EN 1993-1-1:2005).

The NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the \(N_{cr,T,F}\) and \(N_{cr,T}\) factors and therefore these methods are used to evaluate the elastic critical loads for the Polish NA.

The critical axial load for torsional buckling is evaluated as:

\[
i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2\]

where

- \(i_y\) and \(i_z\) = the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

\[
N_{cr,T,F} = \frac{i_o^2}{2(i_y^2 + i_z^2)} \left[ N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4 N_{cr,y} N_{cr,T} \frac{i_y^2 + i_z^2}{i_o^2}} \right]
\]

For details on these equations, refer to the NCCI document SN001a-EN-EU.

**D5.D.9 Singaporean National Annex to EC3**

Note: Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the Singaporean-NA.

The following clauses are not implemented in STAAD.Pro:

**Clause 6.3.2.4(1) B – Slenderness for flexural buckling**

The SINGAPORE NA specifies the value of $\lambda c_0$ for I, H channel or box section to be used in equation 6.59 of SS EN 1993-1-1:2010 as 0.4. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Singapore National Annex.

**Clause 6.3.2.4(2)B – Modification factor “$kfl$”**

The value of the modification factor $kfl$ to be used in equation 6.60 of SS EN 1993-1-1. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Singapore National Annex.

The clauses/sections in EN 1993-1-1:2010 (hereafter referred to as EC-3) that have been dealt with in the Singaporean National Annex (hereafter referred to as SS-NA) and that are relevant to the proposed implementation are:

**D5.D.9.1 Clause 6.1 – General**

The partial safety factors will use the following values:

- Resistance of cross-sections, $\gamma_{M0} = 1.0$
- Resistance of members to instability, $\gamma_{M1} = 1.0$
- Resistance of cross sections to tension, $\gamma_{M2} = 1.1$

The design function in STAAD.Pro sets these values as the default values for the SS-NA (NA 7 is specified).

Note: You can change these values through the GM0, GM1, & GM2 design parameters. Refer to D5.C.6 Design Parameters (on page 1776)

Note: If any of these parameters have been specified by the user as “0”, STAAD.Pro will ignore the specified value and use the default values as given above.

**D5.D.9.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks**

The Singaporean NA recommends the use of Table 6.3 and 6.4 of NF EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor $X_{LT}$, requires the calculation of the “Elastic Critical Buckling Moment”, $M_c$. The Singaporean National Annex does not specify a particular method to calculate $M_c$. Hence the calculation of $M_c$ has been based on the following NCCI documents:

SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling

This document provides a method to calculate “$M_c$” specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method. The equation to evaluate $M_c$ is given in the NCCI as:

$$M_c = C_1 \frac{n^2 EI}{(kL)^2} \left\{ \left( \frac{k}{k_w} \right)^2 \frac{I_w}{T} + \frac{(kL)^2 GI_t}{n^2 EI} + \left( C_2 Z_g \right)^2 - C_2 Z_g \right\}$$

$C_1$ and $C_2$ are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for $C_1$ and $C_2$ for the different cases as given in the tables below.
Table 155: Values of $C_1$ for end moment loading (for k=1)

<table>
<thead>
<tr>
<th>$\psi$</th>
<th>$C_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>+1,00</td>
<td>1,00</td>
</tr>
<tr>
<td>+0,75</td>
<td>1,14</td>
</tr>
<tr>
<td>+0,50</td>
<td>1,31</td>
</tr>
<tr>
<td>+0,25</td>
<td>1,52</td>
</tr>
<tr>
<td>0,00</td>
<td>1,77</td>
</tr>
<tr>
<td>-0,25</td>
<td>2,05</td>
</tr>
<tr>
<td>-0,50</td>
<td>2,33</td>
</tr>
<tr>
<td>-0,75</td>
<td>2,57</td>
</tr>
</tbody>
</table>

This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the $CMM$ parameter.

SN030a-EN-EU – Mono-symmetric uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment ($M_{cr}$) for uniform mono-symmetric sections that are symmetric about the weak axis. Hence, the elastic critical moment for "Tee-Sections" will be evaluated using the method in this NCCI.

**Note:** Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate $M_{cr}$ for mono-symmetric sections is given as:

$$M_{cr} = C_1 \left( \frac{n^2EI_z}{k_xL} \right)^2 \left[ \frac{k_x}{k_w} \frac{2I_w}{T} + \frac{(k_xL)^2GT}{n^2EI_z} + (C_2z_g - C_3z_1)^2 - C_2z_g - C_3z_1 \right]$$

The factors $C_1$, $C_2$ and $C_3$ are dependent on the end conditions and loading criteria. This implementation will consider $C_1$, $C_2$ and $C_3$ as given in the tables below:
The CMM parameter specified during design input will determine the values of C1, C2 and C3. The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the “end moments and transverse loading” condition. The user however can use the new “C1”, ‘C2’ and ‘C3’ parameters to input the required values for C1, C2 and C3 to be used in calculating Mcr.

**Note:** If “MU” as well as C1, C2 and C3 have been specified, the program will ignore MU and use the user input values of C1, C2 and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term "zg" in the equation to calculate Mcr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of ‘zg’ is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The user will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0
in the design input would indicate that the load acts exactly at the shear center of the section so that the term “zg” in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

### D5.D.9.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The Singaporean NA specifies different values for the $\lambda_{LT,0}$ and $\beta$ factors to be used in equation 6.57 of SS EN 1993-1-1 for rolled and equivalent welded sections. STAAD.Pro does not differentiate between rolled and welded sections and uses the default values in SS EN 1993-1-1 for $\lambda_{LT,0}$ and $\beta$. The values specified in the Singapore NA are:

- For rolled sections and hot-rolled & cold formed hollow sections:
  \[
  \lambda_{LT,0} = 0.4 \\
  \beta = 0.75 
  \]

- For welded sections:
  \[
  \lambda_{LT,0} = 0.2 \\
  \beta = 1.00 
  \]

STAAD.Pro uses the buckling curves based on Table 6.5 of SS EN 1993-1-1:2005. The Singaporean-NA provides the values for the terms $\lambda_{LT0}$ and $\beta$ factors given in clause 6.3.2.3(1) as follows:

### Table 156: Buckling curves to use with SS-EN 1993-1-1:2005

<table>
<thead>
<tr>
<th>Cross Section</th>
<th>Limits</th>
<th>Buckling Curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rolled doubly symmetric I and H sections and hot-finished hollow sections</td>
<td>h/b $\leq$ 2</td>
<td>b</td>
</tr>
<tr>
<td></td>
<td>2.0 &lt; h/b $\leq$ 3.1</td>
<td>c</td>
</tr>
<tr>
<td></td>
<td>h/b $&gt;$ 3.1</td>
<td>d</td>
</tr>
<tr>
<td>Angles (for moments in the major principle plane)</td>
<td></td>
<td>d</td>
</tr>
<tr>
<td>All other hot-rolled sections</td>
<td></td>
<td>d</td>
</tr>
<tr>
<td>Welded, doubly symmetric sections and cold-formed hollow sections</td>
<td>h/b $\leq$ 2</td>
<td>c</td>
</tr>
<tr>
<td></td>
<td>2.0 &lt; h/b $\leq$ 3.1</td>
<td>d</td>
</tr>
</tbody>
</table>

Note: This table does not specify which buckling curve is to be used in case of welded doubly symmetric sections with h/b $\geq$ 3.1 and welded non-doubly symmetric sections. Hence for these cases the new implementation will still use the method specified in the base code as per clause 6.3.2.2(2).

### D5.D.9.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, $\chi_{LT}$ as per Singaporean NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005) both give equations to evaluate the LTB reduction factor $\chi_{LT}$ to be used in eqn. 6.55 of SS EN 1993-1-1:2005.
Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating \( \chi_{LT} \). Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of \( \chi_{LT} \) should be determined from...”. Hence in the implementation of EC3 (and the Singaporean Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate \( \chi_{LT} \). For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate \( \chi_{LT} \).

Cl. 6.3.2.3 in the Singaporean National Annex states that Table 6.5 in SS EN 1993-1-1:2005 should be replaced with the table given in the NA (See section 4.3 of this document). Hence for all cases dealt with by the table in the Singaporean NA, this implementation will choose the buckling curves from the Singaporean National Annex. For any case that is not dealt with by the table in the Singaporean NA, the program will use the method given in Cl. 6.3.2.2 of SS EN 1993-1-1:2005.

For the following cross sections, the program will use the Table in the Singaporean NA for choosing a buckling curve for LTB checks (when the SS EN has been specified):

- Rolled doubly symmetric I & H Sections
- Rolled doubly symmetric hollow sections (SHS, RHS, CHS)
- Angle Sections
- Any other rolled section
- Welded doubly symmetric sections with h/b < 3.1

For the following cross sections, the program will use Cl. 6.3.2.3 of SS EN 1993-1-1:2005 to evaluate \( \chi_{LT} \)

- Welded I & H Sections with h/b ≥ 3.1.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate \( \chi_{LT} \).

In any case, the elastic critical moment, \( M_{cr} \) (used to evaluate the non dimensional slenderness) will be evaluated as given above. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the CMN parameter=1.0 (See section above) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

**Note:** If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

**D5.D.9.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks**

The Singaporean NA specifies the use of Equation 6.58 of SS EN 1993-1-1:2005 to evaluate the modification factor “\( f \)” for the LTB reduction factor \( \chi_{LT} \). To evaluate the modification factor SS EN 1993-1-1:2005 uses a correction factor “\( kc \)” given by Table 6.6 in the code.

The Singapore-NA however, specifies that the correction factor “\( kc \)” is to be obtained as below:

\[
K_c = 1 / \sqrt{C_1}
\]

Where:
C₁ is to be obtained from the NCCI documents as previously described (Refer to **D5.D.9.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks** (on page 1832)). The NCCI document SN003a-EN-EU specifies the values of C₁ to be used in table 3.1 as shown below. The current implementation does not account for the Kᵣ factor and conservatively uses a reduction factor equal to 1. The program allows for the reduction factor based on the Singaporean-NA.

These values are for an end restraint factor of k = 1 (i.e., design parameter CMN = 1.0). Hence for all other values of CMN (i.e., 0.7 or 0.5) this implementation will use the values of C₁ from DD ENV 1993-1-1:1992 Annex F.

The program will use a default value of 1.0 for Kᵣ. However, you can also input a custom value of Kᵣ by setting the design parameter KC to the desired value. If the KC parameter in the design input is set to 0, then the program will automatically calculate its value. This will cause the program to evaluate a value of C₁ corresponding to the end conditions and the Bending moment of the member and in turn calculate Kᵣ as given in the NA. To evaluate C₁, the program will use the NCCI documents as previously described.

**D5.D.9.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz**

The Singaporean NA recommends the methods in either Annex A or Annex B of SS-EN 1993-1-1 to calculate these interaction factors. The current implementation of EC3 BS in STAAD.pro uses the method in Annex B by default. The proposed implementation of the Singaporean NA will also use Annex B for Cl.6.3.3 checks.

However for non-doubly symmetric sections, the Singaporean NA gives the option of using Annex B with some modifications as given in the NA. (Cl. NA-3.2 of the Singaporean NA). The Singaporean NA requires additional checks to be done to check for the maximum allowable values of λ and X to be used in equations 6.61 and 6.62 of SS EN 1993-1-1:2005.

As per the Singaporean NA, for non-doubly symmetric sections, the slenderness about the weak axis (λᵧ in STAAD) and the corresponding reduction factor χᵧ should be taken as the values from the highest values of slenderness (λ) among the flexural buckling slenderness (λᵧ), torsional slenderness (λT) and torsional-flexural slenderness (λTF) as given in Clauses 6.3.1.3 and 6.3.1.4 of SS EN 1993-1-1:2005. Hence for non-doubly symmetric sections the program will calculate the critical non-dimensional slenderness as:

\[
\lambda_{y} = \text{the maximum of either } \lambda \text{ from Cl 6.3.1.3 or } \lambda_{T} \text{ from Cl 6.3.1.4}
\]

Where:

\[
\lambda_{T} = \sqrt{\frac{A \cdot f_{y}}{N_{cr}}} \quad N_{cr} = \min \left( N_{crT}, N_{crTF} \right).
\]

The Singaporean NA or EC3 does not, however, specify a method to evaluate N_{crT} or N_{crTF}. Therefore, the program uses the method specified in the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” to calculate these. Refer to **D5.D.9.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural bucking** (on page 1838).

**Note:** The Singaporean National Annex or EC3 does not deal with angle sections in specific and hence this implementation will use the method used in the current EC3 implementation to deal with slenderness of angle sections. In the current implementation this is done as per cl 4.7.10 of BS 5950. This proposed implementation will still use the same method for single and double angle sections to evaluate the slenderness.

Clause NA 3.2 of the Singaporean NA also requires that “Where the section is not an I Section or a hollow section and is a class 1 or class 2 section, it will be treated as a class 3 section for the purposes of this clause”. Hence, for all Class 1 or Class 2 cross sections that are not I, H, SHS, RHS or CHS sections, the elastic properties will be used for the purposes of 6.3.3 checks.
**D5.D.9.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling**

Equations 6.52 and 6.53 of SS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness parameter, $\lambda_T$, to be used for torsional and torsional-flexural buckling checks. The SS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of SS EN 1993-1-1:2005). Therefore, the NCCI document "SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes" provides methods to calculate the $N_{cr,T,F}$ and $N_{cr,T}$ factors and hence will to be included in this implementation of the Singaporean NA.

The program will only consider Channel Sections and Tee- sections while working out the critical torsional and Flexural Torsional buckling loads as per Cl 6.3.1.4.

The critical axial load for torsional buckling is evaluated as:

$$i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2$$

where $i_y$ and $i_z$ = the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively.

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2(i_y^2 + i_z^2)} \left[ N_{cr,y} + N_{cr,T} - \left( N_{cr,y} + N_{cr,T} \right)^2 - 4N_{cr,y}N_{cr,T} \left( i_y^2 + i_z^2 \right) \right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

**D5.D.10 Belgian National Annex to EC3**


The following clauses are **not** implemented in STAAD.Pro:

- **Clause 6.3.2.4(1) B – Slenderness for flexural buckling**
  STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Belgian National Annex.

- **Clause 6.3.2.4(2)B – Modification factor “kfl”**
  STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Belgian National Annex.

**Note:** Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the NBN-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Belgian National Annex (hereafter referred to as NBN-NA) and that are relevant to the proposed implementation are:

**D5.D.10.1 Clause 6.1(1) – General**

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are $\gamma_{M0}$, $\gamma_{M1}$, and $\gamma_{M2}$. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Belgian National Annex:

- Resistance of cross-sections, $\gamma_{M0} = 1.0$
D5.D.10.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The NBN-NA recommends the use of Table 6.3 and 6.4 of EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor $\chi_{LT}$, requires the calculation of the Elastic Critical Buckling Moment, $M_{cr}$. The NBN-NA gives a method to calculate $M_{cr}$ in Annex D, which is used by STAAD.Pro. Annex D, however, only deals with the calculation of $M_{cr}$ for doubly symmetric sections and mono symmetric sections that are symmetric about the minor axis (i.e., Tee sections). For any other type of section that is not dealt with by Annex D, STAAD.Pro uses the method and tables given in Annex F of DD ENV 1993-1-1:1992:

Doubly symmetric sections

Annex D of NBN-NA provides equation used to calculate $M_{cr}$ specifically for doubly symmetric sections:

$$M_{cr} = C_1 \frac{n^2 EI}{(kL)^2} \left[ \left( \frac{k}{k_w} \right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI}{n^2 EI} + \left( C_2 Z_g \right)^2 - C_2 Z_g \right]$$

$C_1$ & $C_2$ are factors that depend on the end conditions and the loading conditions. The Annex provides values for $C_1$ & $C_2$ for the different cases as given in Table1 and Table 2 of the Annex. Table 1 deals with the condition of a simply supported member with end moments and the value of $C_1$ is determined by the end moment ratio (Refer to the NA for details). Clause 3.2 of the National Annex however gives a formula to calculate $C_1$ as:

$$C_1 = 1.77 - 1.04\psi + 0.27\psi^2 \leq 2.60$$

The value of $C_2$ is determined based on the Table 2 of the Annex, based on the loading and end conditions as specified using the $CMM$ parameter.

This NBN-NA considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the $CMM$ parameter.

Mono-symmetric sections with symmetry about their weak axis

Annex D of NBN-NA also provides a method to evaluate the elastic critical moment, $M_{cr}$, for uniform mono symmetric sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for Tee-Sections is evaluated using the method in this Annex.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.
The equation to evaluate $M_{cr}$ for mono symmetric sections is given as:

$$M_{cr} = C_1 \left( \frac{n^2 E I_z}{(k_x L)^2} \right) \left( \left( \frac{k_x}{k_w} \right)^2 \frac{I_w}{I_w} + \frac{(k_x L)^2 G T}{n^2 E I_z} + (C_2 z_g - C_3 z_1)^2 - C_2 z_g - C_3 z_1 \right)$$

The factors $C_1$, $C_2$, and $C_3$ are dependent on the end conditions and loading criteria. This implementation will consider $C_1$, $C_2$ and $C_3$ as given in the tables below:

Table 157: Critical moment coefficients for singly symmetric sections with end moments

<table>
<thead>
<tr>
<th>End Moments and Support Conditions</th>
<th>Bending moment diagram</th>
<th>$k_x$</th>
<th>Value of coefficients</th>
<th>$C_1$</th>
<th>$C_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M$</td>
<td>$\psi = \psi_M$</td>
<td>1.0</td>
<td>$\psi_M \leq 0$</td>
<td>1.0</td>
<td>1.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi_M &gt; 0$</td>
<td>0.5</td>
<td>1.05</td>
</tr>
<tr>
<td>$\psi = +3/4$</td>
<td></td>
<td>1.0</td>
<td>$\psi = +3/4$</td>
<td>1.0</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = +3/4$</td>
<td>0.5</td>
<td>1.19</td>
</tr>
<tr>
<td>$\psi = +1/2$</td>
<td></td>
<td>1.0</td>
<td>$\psi = +1/2$</td>
<td>1.0</td>
<td>1.31</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = +1/2$</td>
<td>0.5</td>
<td>1.37</td>
</tr>
<tr>
<td>$\psi = +1/4$</td>
<td></td>
<td>1.0</td>
<td>$\psi = +1/4$</td>
<td>1.0</td>
<td>1.52</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = +1/4$</td>
<td>0.5</td>
<td>1.60</td>
</tr>
<tr>
<td>$\psi = 0$</td>
<td></td>
<td>1.0</td>
<td>$\psi = 0$</td>
<td>1.0</td>
<td>1.77</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = 0$</td>
<td>0.5</td>
<td>1.86</td>
</tr>
<tr>
<td>$\psi = -1/4$</td>
<td></td>
<td>1.0</td>
<td>$\psi = -1/4$</td>
<td>1.0</td>
<td>2.06</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = -1/4$</td>
<td>0.5</td>
<td>2.15</td>
</tr>
<tr>
<td>$\psi = -1/2$</td>
<td></td>
<td>1.0</td>
<td>$\psi = -1/2$</td>
<td>1.0</td>
<td>2.35</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = -1/2$</td>
<td>0.5</td>
<td>2.42</td>
</tr>
<tr>
<td>$\psi = -3/4$</td>
<td></td>
<td>1.0</td>
<td>$\psi = -3/4$</td>
<td>1.0</td>
<td>2.60</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = -3/4$</td>
<td>0.5</td>
<td>2.45</td>
</tr>
<tr>
<td>$\psi = -1$</td>
<td></td>
<td>1.0</td>
<td>$\psi = -1$</td>
<td>1.0</td>
<td>2.60</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>$\psi = -1$</td>
<td>0.5</td>
<td>2.45</td>
</tr>
</tbody>
</table>
**Note:** According to Section 3(1): \( Czg = 0 \)

### Table 158: Value of coefficients

<table>
<thead>
<tr>
<th>Load and support conditions</th>
<th>Bending moment diagram</th>
<th>( k_z )</th>
<th>Value of coefficients</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>( C_1 ) 1.12, ( C_2 ) 0.45, ( C_3 ) 0.525</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>( C_1 ) 0.97, ( C_2 ) 0.36, ( C_3 ) 0.478</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>( C_1 ) 1.35, ( C_2 ) 0.59, ( C_3 ) 0.411</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
<td>( C_1 ) 1.05, ( C_2 ) 0.48, ( C_3 ) 0.338</td>
</tr>
</tbody>
</table>

The CMM parameter specified during design input will determine the values of \( C_1 \), \( C_2 \), and \( C_3 \). The default value of CMM is 0, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the “end moments and transverse loading” condition. The user however can use the new “\( C_1 \)”, “\( C_2 \)” and “\( C_3 \)” parameters to input the required values for \( C_1 \), \( C_2 \) and \( C_3 \) to be used in calculating \( Mcr \).

**Note:** If “MU” as well as \( C_1 \), \( C_2 \) and \( C_3 \) have been specified, the program will ignore MU and use the user input values of \( C_1 \), \( C_2 \) and \( C_3 \). STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member \( (k = kw = 1.0) \). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of \( K = kw = 1 \) is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints \( (i.e., \text{CMN} = 1.0) \). For members with partial or end fixities \( (i.e., \text{CMN} = 0.5 \text{ or CMN} = 0.7) \), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term “\( zg \)” in the equation to calculate \( Mcr \) refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of “\( zg \)” is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to \( \text{Depth of section} / 2 \) from the shear center. The user will be allowed to modify this value by using the \( ZG \) parameter. Specifying a value of \( ZG = 0 \) in the design input would indicate that the load acts exactly at the shear center of the section so that the term “\( zg \)” in the equation will have a value of zero.

**Note:** The program does not consider the case of cantilevers.
**D5.D.10.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section**

The NBN-NA recommends the use of the values specified in EN 1993-1-1 for the LTB factors $\lambda_{LT0}$ and $\beta$. However, it gives two different sets of values for $\lambda_{LT0}$ & $\beta$ based on two different conditions as given below:

1. If $M_{cr}$ is determined by considering the properties of the gross cross section and the lateral restraints, the following values are used:
   
   $\lambda_{LT0} = 0.2$ and $\beta = 1.0$

2. If $M_{cr}$ is determined by ignoring the lateral restraints, the following values are used:
   
   $\lambda_{LT0} = 0.4$ and $\beta = 0.75$

The program evaluates which factors to use based on the CMN parameter. If CMN = 1.0 (default), then the program assumes the restraints are ignored and the second set of values is used for $\lambda_{LT0}$ and $\beta$. If CMN = 0.5, then the first set of $\lambda_{LT0}$ and $\beta$ values is used.

These factors are then applied to equation 6.57 of NBN-EN to evaluate the Lateral Torsional Buckling reduction factor $\chi_{LT}$.

**D5.D.10.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, $\chi_{LT}$ as per Belgium NA**

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005) both give equations to evaluate the LTB reduction factor $\chi_{LT}$ to be used in eqn. 6.55 of NBN-EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating $\chi_{LT}$. Table 6.4 specifies the choice of buckling curves for “Rolled I Sections”, “Welded I Sections” and “Any other sections”. Cl. 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of $\chi_{LT}$ should be determined from...”. Hence in the implementation of EC3 (and the Belgian Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate $\chi_{LT}$. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

Cl. 6.3.2.3 in the Belgian National Annex gives equations to evaluate the imperfection factors to be used for various section types. (Refer to D5.D.10.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section (on page 1842) ). Hence for all cases dealt with by the equations in the NBN-NA, this implementation will use Cl. 6.3.2.3 to evaluate $\chi_{LT}$.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

In any case, the elastic critical moment $M_{cr}$, (used to evaluate the non dimensional slenderness) will be evaluated as given above. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the CMN parameter=1.0 (Refer to D5.D.10.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section (on page 1842) ) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

You can override the default behavior and specify the clause that is to be used for LTB checks. This can be specified using the MTH design parameter (Refer to D5.C.6 Design Parameters (on page 1776)).

**Note:** If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of NBN-EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).
D5.D.10.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The Belgian NA specifies that the modification factor is to be obtained as per the default method given in EC-3. Hence the proposed implementation will use the existing functionality to work out the correction factor “kc” to be used in the modification factor f.

The program uses a default value of 1.0 for “kc”. However the user can also input a custom value of ‘kc’ by setting the design parameter ‘KC’ to the desired value. The user can also get the program to calculate the value of ‘kc’ automatically by setting the value of the ‘KC’ parameter in the design input to 0. This will cause the program to work out ‘kc’ from table 6.6 of NBN EN 1993-1-1:2005. This will correspond to the end conditions and the bending moment of the member (i.e the value of CMM parameter specified).

- For CMM = 7 the program will choose the value of “kc” to be either 0.90 or 0.91 based on the end moment ratio.
- For CMM = 8 the program will choose the value of “kc” to be either 0.77 or 0.82 based on the end moment ratio.

An additional check will also be performed as given below:

\[ \chi_{LT,\text{mod}} \leq \frac{1}{\chi_{LT}^2} \]

D5.D.10.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The NBN-NA recommends the equations in Annex A of NBN-EN 1993-1-1 to calculate these interaction factors. The NA also mentions that torsional flexural buckling needs to be taken into account in case of mono symmetric sections. Torsional flexural buckling will need to be taken into account based on the method given in the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes”. See section below for details.

The NA also recommends a lower limit as given below for the term \( C_{mi,0} \) in table A.2 of Annex A:

\[ C_{mi,0} \geq 1 - \frac{N_{Ed}}{N_{cr,i}} \]

D5.D.10.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of NBN-EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness parameter, \( \lambda_T \), to be used for torsional and torsional-flexural buckling checks. The NBN-EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads \( N_{cr,T,F} \) and \( N_{cr,T} \) (refer 6.3.14 of SS EN 1993-1-1:2005).

Therefore, the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the \( N_{cr,T,F} \) and \( N_{cr,T} \) factors and hence will to be included in this implementation of the Belgian NA.

The critical axial load for torsional buckling is evaluated as:

\[
\begin{align*}
    i_y^2 & = i_y^2 + i_z^2 + y_o^2 + z_o^2 \\
    i_y \text{ and } i_z & \text{ the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively}
\end{align*}
\]

The critical axial load for torsional-flexural buckling is evaluated as:
\[
N_{cr,TF} = \frac{i_0^2}{2(i_y^2 + i_z^2)} \left[ N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4 N_{cr,y} N_{cr,T} \frac{i_y^2 + i_z^2}{i_0^2}} \right]
\]

For details on these equations, refer to the NCCI document SN001a-EN-EU.

The program will only consider Channel Sections and Tee- sections while working out the critical torsional and Flexural Torsional buckling loads as per Cl 6.3.1.4.

**D5.D.11 Malaysian National Annex to EC3**


The following clauses are *not* implemented in STAAD.Pro:

- **Clause 6.3.2.4(1) B – Slenderness for flexural buckling**
  
  STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Malaysian National Annex.

- **Clause 6.3.2.4(2)B – Modification factor “kfl”**
  
  STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Malaysian National Annex.

**Note:** Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the MS-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Malaysian National Annex (hereafter referred to as MS-NA) and that are relevant to the proposed implementation are:


EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are \( \gamma_{M0}, \gamma_{M1}, \) and \( \gamma_{M2} \). EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Malaysian National Annex:

- Resistance of cross-sections, \( \gamma_{M0} = 1.0 \)
- Resistance of members to instability, \( \gamma_{M1} = 1.0 \)
- Resistance of cross sections to tension, \( \gamma_{M2} = 1.1 \)

The design function in STAAD.Pro sets these values as the default values for the MS-NA (NA 9 is specified).

**Note:** You can change these values through the \( GM0, GM1, \) & \( GM2 \) design parameters. Refer to D5.C.6 Design Parameters (on page 1776)

**Note:** If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

**D5.D.11.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks**

The MS-NA recommends the use of Table 6.3 and 6.4 of MS EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.
The calculation of the LTB reduction factor $\chi_{LT}$ requires the calculation of the Elastic Critical Buckling Moment, $M_{cr}$. The MS-NA does not specify a particular method to calculate $M_{cr}$. Hence the calculation of $M_{cr}$ has been based on the following NCCI documents:

Doubly symmetric sections

SN003a-EN-EU NCCI: Elastic critical moment for lateral torsional buckling provides equation used to calculate $M_{cr}$ specifically for doubly symmetric sections:

$$M_{cr} = C_1 \pi^2 E I \left( \frac{k L}{k w} \right)^2 \bar{I}_w + \frac{(kL)^2 G I_t}{n^2 E I_s} + (C_2 Z_g - C_3 z_1)^2 - C_2 Z_g$$

$C_1$ and $C_2$ are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for $C_1$ and $C_2$ for the different cases as given in Table 3.1 and Table 3.2.

The NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the CMM parameter. The values of $C_1$ and $C_2$ may also be directly specified using the $C1$ and $C2$ parameters, respectively (required for $CMM = 7$ or $CMM = 8$).

Mono-symmetric sections with symmetry about their weak axis

Annex D of MS-NA also provides a method to evaluate the elastic critical moment, $M_{cr}$, for uniform mono-symmetric sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for Tee-Sections is evaluated using the method in this Annex.

**Note:** Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate $M_{cr}$ for mono symmetric sections is given as:

$$M_{cr} = C_1 \left( \frac{k L}{k w} \right)^2 \left( \frac{I_w}{T_S} + \frac{(kL)^2 G I_t}{n^2 E I_s} + (C_2 Z_g - C_3 z_1)^2 - C_2 Z_g - C_3 z_1 \right)$$

The factors $C_1$, $C_2$, and $C_3$ are dependent on the end conditions and loading criteria. The program considers $C_1$, $C_2$, and $C_3$ as given in the tables 4.1 and 4.2 of the NCCI, based on the CMM parameter.

The default value of $CMM = 0$, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the “end moments and transverse loading” condition. You use the $C1$, $C2$ and $C3$ parameters to input the required values for $C_1$, $C_2$, and $C_3$, respectively, to be used in calculating $M_{cr}$.

**Note:** If “MU” as well as $C1$, $C2$ and $C3$ have been specified, the program will ignore MU and use the user input values of $C1$, $C2$ and $C3$. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

**Note:** When $CMM = 7$ or $CMM = 8$, the values for $C1$, $C2$ and $C3$ parameters must be manually specified.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member ($k = kw = 1.0$). STAAD.Pro takes into account of...
the end conditions using the CMN parameter for EC3. A value of K = kw = 1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term \( z_g \) in the equation to calculate \( M_{cr} \) refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of \( z_g \) is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to \((\text{Depth of section}/2)\) from the shear center. The use will be allowed to modify this value by using the \( ZG \) parameter. Specifying a value of \( ZG = 0 \) in the design input would indicate that the load acts exactly at the shear center of the section so that the term \( z_g \) in the equation will have a value of zero.

**Note:** The program does not consider the case of cantilevers.

**D5.D.11.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section**

The MS-NA specifies different values for the \( \lambda_{LT,0} \) and \( \beta \) factors to be used in equation 6.57 of MS EN 1993-1-1 for rolled and equivalent welded sections. STAAD.Pro does not differentiate between rolled and welded sections and uses the default values in MS EN 1993-1-1 for \( \lambda_{LT,0} \) and \( \beta \). The values specified in the MS-NA are:

1. For rolled sections and hot-rolled & cold formed hollow sections:
   \[ \lambda_{LT,0} = 0.4 \text{ and } \beta = 0.75 \]
2. For welded sections:
   \[ \lambda_{LT,0} = 0.2 \text{ and } \beta = 1.00 \]

STAAD.Pro uses the buckling curves based on Table 6.5 of MS EN 1993-1-1:2005, based on different limits. This table again does not specify which buckling curve is to be used in case of welded doubly symmetric sections with \( h/b \geq 3.1 \) and welded non-doubly symmetric sections. Hence for these cases the new implementation will still use the method specified in the base code as per clause 6.3.2.2(2).

**D5.D.11.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, \( \chi_{LT} \) as per Malaysian NA**

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor \( \chi_{LT} \) to be used in eqn. 6.55 of MS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating \( \chi_{LT} \). Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".

Cl. 6.3.2.2 states "Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of \( \chi_{LT} \) should be determined from...". Hence in the implementation of EC3 (and the MS NA) in STAAD.Pro, by default the program will consider clause Cl. 6.3.2.3 to evaluate \( \chi_{LT} \). For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate \( \chi_{LT} \).

Cl. 6.3.2.3 in the MS NA states that Table 6.5 in MS EN 1993-1-1:2005 should be replaced with the table given in the NA (Refer to **D5.D.11.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section** (on page 1846)). Hence for all cases dealt with by the table in the MS NA, this implementation will choose the buckling
curves from the MS NA. For any case that is not dealt with by the table in the MS NA, the program will use the method given in Cl. 6.3.2.2 of MS EN 1993-1-1:2005.

Hence for the following cross sections the program will use the Table in the MS NA for choosing a buckling curve for LTB checks (when the MS NA has been specified):

- Rolled doubly symmetric I & H Sections
- Rolled doubly symmetric hollow sections (SHS, RHS, CHS)
- Angle Sections
- Any other rolled section
- Welded doubly symmetric sections with $h/b < 3.1$

For the following cross sections, the program will use Cl. 6.3.2.3 of MS EN 1993-1-1:2005 to evaluate $\chi_{LT}$

- Welded I & H Sections with $h/b \geq 3.1$.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

In any case the Elastic critical moment "Mcr" (used to evaluate the non dimensional slenderness) will be evaluated as described in "Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks" (on page 1844). Since the MS NA uses the NCCI documents mentioned in the sections above, this implementation will only consider end restraint conditions corresponding to the CMN parameter=1.0. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

**Note:** If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of MS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

**D5.D.11.5 Clause 6.3.2.3(2) – Modification factor, $f$, for LTB checks**

The MS NA specifies the use of eqn. 6.58 of MS EN 1993-1-1:2005 to evaluate the modification factor, $f$, for the LTB reduction factor $\chi_{LT}$. To evaluate the modification factor MS EN 1993-1-1:2005 uses a correction factor, $k_c$, given by Table 6.6 in the code.

The program does not calculate the $k_c$ factor and conservatively uses a reduction factor equal to 1. The proposed implementation will allow for the reduction factor based on the MS NA.

These values are for an end restraint factor of $k = 1$ (i.e., CMN = 1.0). Hence for all other values of CMN (i.e., 0.7 or 0.5), the program uses the values of $C_1$ from DD ENV 1993-1-1:1992 Annex F.

You can also manually specify a value for $k_c$ by setting the design parameter, $KC$, to the desired value. The user can also get the program to calculate the value of $k_c$ automatically by setting the value of the $KC$ parameter in the design input to 0. This will cause the program to evaluate a value of $C_1$ corresponding to the end conditions and the Bending moment of the member and then calculate $k_c$ as given in the NA. To evaluate $C_1$, the program will use the NCCI documents (Refer to **D5.D.11.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks** (on page 1844)).

Note that for the MS NA, the program will attempt to evaluate $k_c$ by default using the equation in NA,

$$k_c = 1 / \sqrt{C_1}$$

where $C_1$ will be the value used for the $M_{cr}$ calculations.

If $k_c$ evaluates to be greater than 1.0, the program will then evaluate $k_c$ as per Table 6.6 of EN 1993-1-1:2005.
D5.D.11.6 Clause 6.3.3(5) – Interaction factors $k_{yy}$, $k_{yz}$, $k_{zy}$, and $k_{zz}$

The MS NA recommends that the method in Annex A or Annex B of MS EN 1993-1-1:2005 can be used to calculate the interaction factors for Cl. 6.3.3 checks in the case of doubly symmetric sections. STAAD.Pro uses the equations in Annex B of MS EN 1993-1-1:2005 to calculate these interaction factors for doubly symmetric sections.

However, for non-doubly symmetric sections, the MS NA gives the option of using Annex B with some modifications as given in the NA. (Cl. NA-3.2 of the MS NA). The MS NA requires additional checks to be done to check for the maximum allowable values of $\lambda$ and $X$ to be used in equations 6.61 and 6.62 of MS EN 1993-1-1:2005.

As per the MS NA, for non-doubly symmetric sections, the slenderness about the weak axis ($\lambda_y$ in STAAD.Pro) and the corresponding reduction factor $\chi_y$ should be taken as the values from the highest values of slenderness ($\lambda$) among the flexural buckling slenderness ($\lambda_y$), torsional slenderness ($\lambda_T$) and torsional-flexural slenderness ($\lambda_{TF}$) as given in Clauses 6.3.1.3 and 6.3.1.4 of MS EN 1993-1-1:2005. Hence for non-doubly symmetric sections the program will calculate the critical non-dimensional slenderness as:

$$\lambda_y = \max \left\{ \lambda \text{ per Cl. 6.3.1.3}, \lambda_T \text{ per Cl. 6.3.1.4} \right\}$$

where

$$\lambda_T = \sqrt{\frac{A \cdot f_y}{N_{cr}}}$$

$$N_{cr} = \min (N_{crT}, N_{crTF})$$

The MS NA or EC3 does not, however, specify a method to evaluate $N_{crT}$ or $N_{crTF}$. Hence, the program uses the method specified in the NCCI document SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes to calculate these. Refer to D5.D.11.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling (on page 1848) for details.

Note: The MS NA or EC3 does not deal with angle sections specifically and therefore STAAD.Pro uses the method described in the EC3 implementation to deal with slenderness of angle sections. This is done as per cl 4.7.10 of BS 5950.

Clause NA 3.2 of the MS NA also requires that "Where the section is not an I Section or a hollow section and is a class1 or class 2 section, it will be treated as a class 3 section for the purposes of this clause". Hence for all Class 1 or Class 2 cross sections that are not I, H, SHS, RHS or CHS sections, the elastic properties will be used for the purposes of 6.3.3 checks.

D5.D.11.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of MS-EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness parameter, $\lambda_T$, to be used for torsional and torsional-flexural bucking checks. The MS-EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{crT,F}$ and $N_{cr,T}$. (refer 6.3.14 of SS EN 1993-1-1:2005). Therefore, the NCCI document SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes provides methods to calculate the $N_{crT,F}$ and $N_{cr,T}$ factors and hence will to be included in this implementation of the MS NA.

The program will only consider Channel Sections and Tee- sections when evaluating the critical torsional and Flexural Torsional buckling loads as per Cl 6.3.1.4.

The critical axial load for torsional buckling is evaluated as:
where
\[ i_o^2 = i_y^2 + i_z^2 + y_0^2 + z_0^2 \]
\[ i_y \text{ and } i_z = \text{the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively} \]

The critical axial load for torsional-flexural buckling is evaluated as:
\[ N_{cr,TF} = \frac{i_o^2}{2(i_y^2 + i_z^2)} \left[ N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}} \right] \]

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.12 German National Annex to EC3


The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that require additional clauses from the German National Annex (hereafter referred to as DE-NA) are described in the following sections.

Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the DE-NA.

The following clauses are not implemented in STAAD.Pro:

- **Clause 6.3.2.4(1) B – Slenderness for flexural buckling**
  STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the German National Annex.

- **Clause 6.3.2.4(2)B – Modification factor “kfl”**
  STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the German National Annex.


EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are \( \Gamma_{M0} \), \( \Gamma_{M1} \), and \( \Gamma_{M2} \). EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the German National Annex:

- Resistance of cross-sections, \( \Gamma_{M0} = 1.0 \)
- Resistance of members to instability, \( \Gamma_{M1} = 1.0 \)
- Resistance of cross sections to tension, \( \Gamma_{M2} = 1.25 \)

The design function in STAAD.Pro sets these values as the default values for the DE-NA (NA 10 is specified).

**Note:** You can change these values through the GM0, GM1, & GM2 design parameters. Refer to **D5.C.6 Design Parameters** (on page 1776)

**Note:** If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.
D5.D.12.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks

The DE-NA recommends the use of Table 6.3 and 6.4 of DIN EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor $\chi_{LT}$, requires the calculation of the Elastic Critical Buckling Moment, $M_{cr}$. The DE-NA does not specify a particular method to calculate $M_{cr}$. Hence the calculation of $M_{cr}$ has been based on the following NCCI documents:

Doubly symmetric sections

SN003a-EN-EU NCCI: Elastic critical moment for lateral torsional buckling provides equation used to calculate $M_{cr}$ specifically for doubly symmetric sections:

$$M_{cr} = C_1 \frac{n^2EI}{(kL)^2} \left( \frac{kLw}{TS} + \frac{(kL)^2GI_T}{n^2EI_S} + \left(C_2Z_g - C_3Z_1\right)^2 - C_2Z_g - C_3Z_1\right)$$

The factors $C_1$ and $C_2$ are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for $C_1$ and $C_2$ for the different cases as given in Table 3.1 and Table 3.2.

The NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the $CMM$ parameter. The values of $C_1$ and $C_2$ may also be directly specified using the $C1$ and $C2$ parameters, respectively (required for $CMM = 7$ or $CMM = 8$).

Mono-symmetric sections with symmetry about their weak axis

Annex D of DE-NA also provides a method to evaluate the elastic critical moment, $M_{cr}$, for uniform mono-symmetric sections that are symmetric about the weak axis. STAAD.Pro uses this method for the evaluating the elastic critical moment for Tee sections.

The equation to evaluate $M_{cr}$ for mono symmetric sections is given as:

$$M_{cr} = C_1 \frac{n^2EI_z}{(kL)^2} \left( \frac{kLw}{T_T} + \frac{(kL)^2GI_T}{n^2EI_z} + \left(C_2z_g - C_3z_1\right)^2 - C_2z_g - C_3z_1\right)$$

The factors $C_1$, $C_2$, and $C_3$ are dependent on the end conditions and loading criteria. The program considers $C_1$, $C_2$, and $C_3$ as given in the tables 4.1 and 4.2 of the NCCI, based on the $CMM$ parameter.

The default value of $CMM = 0$, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the "end moments and transverse loading" condition. You use the $C1$, $C2$ and $C3$ parameters to input the required values for $C_1$, $C_2$, and $C_3$, respectively, to be used in calculating $M_{cr}$.

**Note:** If "MU" as well as $C1$, $C2$ and $C3$ have been specified, the program will ignore MU and use the user input values of $C1$, $C2$ and $C3$. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

**Note:** When $CMM = 7$ or $CMM = 8$, the values for $C1$, $C2$ and $C3$ parameters must be manually specified.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member ($k = kw = 1.0$). STAAD.Pro takes into account of...
the end conditions using the CMN parameter for EC3. A value of $K = k_w = 1$ is indicated by a value of $CMN = 1.0$ in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term $z_g$ in the equation to calculate $M_{cr}$ refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of $z_g$ is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the $ZG$ parameter. Specifying a value of $ZG = 0$ in the design input would indicate that the load acts exactly at the shear center of the section so that the term $z_g$ in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

**D5.D.12.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section**

The DE-NA specifies that the default values in Table 6.5 for the $\lambda_{LT,0}$ and $\beta$ factors given in clause 6.3.2.3(1) as follows:

For all sections, use:

$$\lambda_{LT,0} = 0.4 \text{ and } \beta = 0.75$$

**D5.D.12.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, $\chi_{LT}$ as per German NA**

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor $\chi_{LT}$ to be used in eqn. 6.55 of DIN EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating $\chi_{LT}$. Table 6.4 specifies the choice of buckling curves for “Rolled I Sections”, “Welded I Sections” and “Any other sections”. Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of $\chi_{LT}$ should be determined from...”. Hence in the implementation of EC3 (and the German Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate $\chi_{LT}$. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate $\chi_{LT}$.

Note: The MTH design parameter can be used to control the choice of the clause used to calculate $\chi_{LT}$.

In any case, the elastic critical moment, $M_{cr}$, used to evaluate the non dimensional slenderness will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the CMN parameter = 1.0 (Refer to **D5.D.12.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks** on page 1850 ) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2
of DIN EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

**D5.D.12.5 Clause 6.3.2.3(2) – Modification factor, \( f \), for LTB checks**

The DE-NA specifies the use of Equation 6.58 of DIN EN 1993-1-1:2005 to evaluate the modification factor “\( f \)” for the LTB reduction factor \( \chi_{LT} \). To evaluate the modification factor SS EN 1993-1-1:2005 uses a correction factor “\( kc \)” given by Table 6.6 in the code.

The DE-NA however, specifies that the correction factor “\( kc \)” is to be obtained as below:

\[
K_c = \frac{1}{\sqrt{C_1}}
\]

Where:

- \( C_1 \) is to be obtained from the NCCI documents as previously described (Refer to **D5.D.12.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks** (on page 1850)). The NCCI document SN003a-EN-EU specifies the values of \( C_1 \) to be used in table 3.1 as shown below. The current implementation does not account for the \( K_c \) factor and conservatively uses a reduction factor equal to 1. The program allows for the reduction factor based on the DE-NA.

These values are for an end restraint factor of \( k = 1 \) (i.e., design parameter \( CMN = 1.0 \)). Hence for all other values of \( CMN \) (i.e., 0.7 or 0.5) this implementation will use the values of \( C_1 \) from DD ENV 1993-1-1:1992 Annex F.

The program will use a default value of 1.0 for \( K_c \). However, you can also input a custom value of \( K_c \) by setting the design parameter \( KC \) to the desired value. If the \( KC \) parameter in the design input is set to 0, then the program will automatically calculate its value. This will cause the program to evaluate a value of \( C_1 \) corresponding to the end conditions and the Bending moment of the member and in turn calculate \( K_c \) as given in the NA. To evaluate \( C_1 \), the program will use the NCCI documents as previously described.

**D5.D.12.6 Clause 6.3.3(5) – Interaction factors \( k_{yy} \), \( k_{yz} \), \( k_{zy} \), and \( k_{zz} \)**

The DE-NA recommends the use of equations in Annex A or Annex B of DIN-EN 1993-1-1 to calculate these interaction factors. STAAD.Pro uses the method in Annex B by default. Thus the program uses Annex B for Cl. 6.3.3 checks.

The DE-NA or EC3 do not deal with angle sections in specific and thus the program uses the method per cl 4.7.10 of BS 5950 for single and double angle sections to evaluate the slenderness.

**D5.D.12.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling**

Equations 6.52 and 6.53 of DIN EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness \( \lambda_T \), to be used for torsional and torsional-flexural buckling checks. DIN EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads \( N_{cr,T,F} \) and \( N_{cr,T} \) (refer 6.3.14 of DIN EN 1993-1-1:2005).

The NCCI document SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes provides methods to calculate the \( N_{cr,T,F} \) and \( N_{cr,T} \) factors and therefore these methods are used to evaluate the elastic critical loads for the DE-NA.

The critical axial load for torsional buckling is evaluated as:

\[
i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2
\]
\[ i_y \text{ and } i_z = \text{the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively} \]

The critical axial load for torsional-flexural buckling is evaluated as:

\[
N_{cr,TF} = \frac{i_o^2}{2(i_y^2 + i_z^2)} \left[ N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}i_o^2(i_y^2 + i_z^2)} \right]
\]

For details on these equations, refer to the NCCI document SN001a-EN-EU.

**D5.D.13 Swedish National Annex to EC3**


The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that require additional clauses from the Swedish National Annex (hereafter referred to as SW-NA) are described in the following sections.

Refer to the basic code (EC3) (on page 1754) for a description of these clauses. The sections below refer to the corresponding clauses in the SW-NA.

**D5.D.13.1 Clause 3.2.(2) - Steel Grades**

The Swedish NA allows the use of custom steel grades as given in Table E-1 of the National Annex. These grades of steel can be specified by using the PY (Yield Strength) and FU (Ultimate Strength) parameters in STAAD.Pro.


EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are \( \Gamma_{M0}, \Gamma_{M1}, \text{ and } \Gamma_{M2} \). EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Swedish National Annex:

- Resistance of cross-sections, \( \Gamma_{M0} = 1.0 \)
- Resistance of members to instability, \( \Gamma_{M1} = 1.0 \)
- Resistance of cross sections to tension, \( \Gamma_{M2} = 0.9 x f_u/f_y \), but not less than 1.1

Where:

- \( f_u \) is the ultimate steel strength
- \( f_y \) is the yield strength of steel

The design function in STAAD.Pro sets these values as the default values for the SW-NA (NA 10 is specified).

**Note:** You can change these values through the GM0 and GM1 design parameters. Refer to D5.C.6 Design Parameters (on page 1776) The value of GM2 (\( \Gamma_{M2} \)) is calculated based on the steel grade values specified. Refer to D5.D.13.1 Clause 3.2.(2) - Steel Grades (on page 1853)

**Note:** If any of these parameters are specified as 0, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.
D5.D.13.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The SW-NA recommends the use of Table 6.3 and 6.4 of DIN EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor $\chi_{LT}$, requires the calculation of the Elastic Critical Buckling Moment, $M_{cr}$. The SW-NA does not specify a particular method to calculate $M_{cr}$. Hence the calculation of $M_{cr}$ has been based on the following NCCI documents:

Doubly symmetric sections

SN003a-EN-EU NCCI: Elastic critical moment for lateral torsional buckling provides equation used to calculate $M_{cr}$ specifically for doubly symmetric sections:

$$M_{cr} = C_1 \frac{n^2 EI}{(kL)^2} \left( \left( \frac{k}{k_w} \right)^2 \frac{I_w}{T} + \frac{(kL)^2 GI_T}{n^2 EI_S} + \left( C_2 Z_g - C_3 Z_1 \right)^2 - C_2 Z_g \right)$$

$C_1$ and $C_2$ are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for $C_1$ and $C_2$ for the different cases as given in Table 3.1 and Table 3.2.

The NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the $CMM$ parameter. The values of $C_1$ and $C_2$ may also be directly specified using the $C1$ and $C2$ parameters, respectively (required for $CMM = 7$ or $CMM = 8$).

Mono-symmetric sections with symmetry about their weak axis

Annex D of SW-NA also provides a method to evaluate the elastic critical moment, $M_{cr}$, for uniform mono-symmetric sections that are symmetric about the weak axis. STAAD.Pro uses this method for the evaluating the elastic critical moment for Tee sections.

The equation to evaluate $M_{cr}$ for mono symmetric sections is given as:

$$M_{cr} = C_1 \frac{n^2 EI_z}{(k_x L)^2} \left( \left( \frac{k_x}{k_w} \right)^2 \frac{I_w}{T} + \frac{(k_x L)^2 GI_T}{n^2 EI_z} + \left( C_2 Z_g - C_3 Z_1 \right)^2 - C_2 Z_g \right)$$

The factors $C_1$, $C_2$, and $C_3$ are dependent on the end conditions and loading criteria. The program considers $C_1$, $C_2$, and $C_3$ as given in the tables 4.1 and 4.2 of the NCCI, based on the $CMM$ parameter.

The default value of $CMM = 0$, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the “end moments and transverse loading” condition. You use the $C1$, $C2$ and $C3$ parameters to input the required values for $C_1$, $C_2$, and $C_3$, respectively, to be used in calculating $M_{cr}$.

**Note:** If "MU" as well as $C1$, $C2$ and $C3$ have been specified, the program will ignore MU and use the user input values of $C1$, $C2$ and $C3$. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

**Note:** When $CMM = 7$ or $CMM = 8$, the values for $C1$, $C2$ and $C3$ parameters must be manually specified.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member ($k = kw = 1.0$). STAAD.Pro takes into account of
the end conditions using the CMN parameter for EC3. A value of \( K = kw = 1 \) is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term \( z_g \) in the equation to calculate \( M_{cr} \) refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of \( z_g \) is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to \((\text{Depth of section}/2)\) from the shear center. The use will be allowed to modify this value by using the \( ZG \) parameter. Specifying a value of \( ZG = 0 \) in the design input would indicate that the load acts exactly at the shear center of the section so that the term \( z_g \) in the equation will have a value of zero.

**Note:** The program does not consider the case of cantilevers.

**D5.D.13.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section**

The SW-NA specifies that the following values for the \( \lambda_{LT,0} \) and \( \beta \) factors:

For all sections, use:

\[
\lambda_{LT,0} = 0.4 \text{ and } \beta = 0.75
\]

**D5.D.13.5 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, \( \chi_{LT} \) as per Swedish NA**

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor \( \chi_{LT} \) to be used in eqn. 6.55 of BFS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating \( \chi_{LT} \). Table 6.4 specifies the choice of buckling curves for “Rolled I Sections”, “Welded I Sections” and “Any other sections”. Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of \( \chi_{LT} \) should be determined from...”. Hence in the implementation of EC3 (and the Swedish Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate \( \chi_{LT} \). For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate \( \chi_{LT} \).

**Note:** The MTH design parameter can be used to control the choice of the clause used to calculate \( \chi_{LT} \).

In any case, the elastic critical moment, \( Mcr \), (used to evaluate the non dimensional slenderness) will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the CMN parameter=1.0 (Refer to D5.D.13.3 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks (on page 1854) ) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

**Note:** If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2
of BFS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

D5.D.13.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The SW-NA recommends the use of equations in Annex A of EN 1993-1-1:2005 to calculate these interaction factors, which are used by STAAD.Pro when the Swedish NA is selected.

The SW-NA or EC3 do not deal with angle sections in specific and thus the program uses the method per cl 4.7.10 of BS 5950 for single and double angle sections to evaluate the slenderness.

D5.D.13.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of BFS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness \( \lambda_T \) to be used for torsional and torsional-flexural buckling checks. BFS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads \( N_{cr,T,F} \) and \( N_{cr,T} \) (refer 6.3.14 of BFS EN 1993-1-1:2005).

The NCCI document SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes provides methods to calculate the \( N_{cr,T,F} \) and \( N_{cr,T} \) factors and therefore these methods are used to evaluate the elastic critical loads for the SW-NA.

The critical axial load for torsional buckling is evaluated as:

\[
i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2
\]

where

\( i_y \) and \( i_z \) = the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

\[
N_{cr,TF} = \frac{i_o^2}{2(N_{cr,y}^2 + N_{cr,z}^2)} \left[ N_{cr,y} + N_{cr,z} - \sqrt{(N_{cr,y} + N_{cr,z})^2 - 4N_{cr,y}N_{cr,z} \frac{i_y^2 + i_z^2}{i_o^2}} \right]
\]

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.E. European Codes - Timber Design Per EC 5: Part 1-1


D5.E.1 General Comments

Principles of Limit States Design of Timber Structures are used as specified in the code.

Design per EC5 is limited to the prismatic, rectangular shapes only. There is no Eurocode-specific timber section database / library consisting of pre-defined shapes for analysis or for design. The feature of member selection is thus not applicable to this code.

The design philosophy of this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load
and resistance factors are used so that a uniform reliability is achieved for all timber structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD implementation of EC 5. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

**Axes convention in STAAD and EC5**

STAAD defines the major axis of the cross-section as zz and the minor axis as yy. The longitudinal axis of the member is defined as x and joins the start joint of the member to the end with the same positive direction.

EC5, however, defines the principal cross-section axes in reverse to that of STAAD, but the longitudinal axis is defined in the same way. Both of these axes definitions follow the orthogonal right hand rule.

![Figure 182: Axis conventions per STAAD and Eurocode 5](image)

**Determination of Factors**

A. $K_{mod}$ – Modification factor taking into account of Load-duration (LDC) and Moisture-content (Service Class - SCL). Reference Table 3.1 of EC-5-2004.

For “Solid Timber”, the values are incorporated in the program.

B. $\gamma_m$ – Partial factor for Material Property values. Reference Table 2.3 of EC-5-2004.

For “Solid Timber”, the value of $\gamma_m = 1.3$ is incorporated in the program.

C. $K_h$ – Size Factor.
For members, subjected to tension, whose maximum c/s dimension is less than the reference width in tension the characteristic strength in tension (f_{0k}) is to be increased by the factor Kh.

For members, subjected to bending, whose depth is less than reference depth in bending, the characteristic strength in bending (f_{mk}) is to be increased by the factor Kh.

As per clause 3.2(3) of EC 5-2004, for rectangular solid timber with a characteristic timber density $\rho_k \leq 700 \text{ kg/m}^3$ the reference depth in bending or the reference width (maximum cross-sectional dimension) is 150 mm.

The value of $Kh = \text{Minimum of } \{(150/h) 0.2 \text{ and } 1.3\}$ for such solid timber is incorporated in the software. Please refer clause numbers 3.3 and 3.4 for the value of Kh for Glued laminated timber and Laminated veneer lumber respectively.

D. KC90 – Factor taking into account the load configuration, possibility of splitting and degree of compressive deformation.

For members, subjected to compression, perpendicular to the direction of grain alignment, this factor should be taken into account. Default value of 1 is used in STAAD.Pro. User may override the value. Please refer clause 6.1.5 of EC-5-2004 in this regard.

E. Km – Factor considering re-distribution of bending stress in cross section.

For members, subjected to bending, this factor is taken into account for stress checking. For rectangular section the value of Km is 0.7, and this value is incorporated in STAAD.Pro. User may override the value. Please refer clause 6.1.6 of EC-5-2004 in this regard.

F. Kshape – Factor depending on shape of cross section.

For members, subjected to torsional force, design torsional stress should be less than equal design shear strength multiplied by the factor Kshape. This factor is determined by STAAD.Pro internally using the guidelines of clause 6.1.8 of EC-5-2004.

### D5.E.2 Analysis Methodology

#### Table 159: EC5 Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_{t0d}$</td>
<td>Design tensile stress parallel (at zero degree) to grain alignment.</td>
</tr>
<tr>
<td>$S_{t90d}$</td>
<td>Design tensile stress perpendicular (at 90 degrees) to grain alignment.</td>
</tr>
<tr>
<td>$S_{c0d}$</td>
<td>Design compressive stress parallel to grain alignment.</td>
</tr>
<tr>
<td>$S_{c90d}$</td>
<td>Design compressive stress perpendicular to grain alignment.</td>
</tr>
<tr>
<td>$S_{mzd}$</td>
<td>Design bending stress about zz axis.</td>
</tr>
<tr>
<td>$S_{myd}$</td>
<td>Design bending stress about yy axis.</td>
</tr>
<tr>
<td>$S_{vd}$</td>
<td>Design shear stress.</td>
</tr>
</tbody>
</table>
### Equations for Characteristic Values of Timber Species as per Annex-A of EN 338:2003

The following equations were used to determine the characteristic values:

#### Design Codes

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_{\text{tor,d}}$</td>
<td>Design torsional stress.</td>
</tr>
<tr>
<td>$F_{\text{t0d}}$</td>
<td>Design tensile strength - parallel to the grain alignment.</td>
</tr>
<tr>
<td>$F_{\text{t90d}}$</td>
<td>Design tensile strength - perpendicular to the grain alignment.</td>
</tr>
<tr>
<td>$F_{\text{c0d}}$</td>
<td>Design compressive strength - parallel to the grain alignment.</td>
</tr>
<tr>
<td>$F_{\text{c90d}}$</td>
<td>Design compressive strength - perpendicular to the grain alignment.</td>
</tr>
<tr>
<td>$F_{\text{mzd}}$</td>
<td>Design bending strength - about zz-axis.</td>
</tr>
<tr>
<td>$F_{\text{myd}}$</td>
<td>Design bending strength - about yy-axis.</td>
</tr>
<tr>
<td>$F_{\text{vd}}$</td>
<td>Design shear strength about yy axis.</td>
</tr>
<tr>
<td>RATIO</td>
<td>Permissible ratio of stresses as input using the RATIO parameter. The default value is 1.</td>
</tr>
<tr>
<td>$l_{z,l_{\text{rel,z}}}$</td>
<td>Slenderness ratios corresponding to bending about zz axis.</td>
</tr>
<tr>
<td>$l_{y,l_{\text{rel,y}}}$</td>
<td>Slenderness ratios corresponding to bending about yy axis.</td>
</tr>
<tr>
<td>$E_{0.05}$</td>
<td>Fifth percentile value of modulus of elasticity parallel to grain.</td>
</tr>
<tr>
<td>$G_{0.05}$</td>
<td>Fifth percentile value of shear modulus parallel to grain.</td>
</tr>
<tr>
<td>$I_z$</td>
<td>Second moment of area about the strong z-axis.</td>
</tr>
<tr>
<td>$I_y$</td>
<td>Second moment of area about the weak y-axis.</td>
</tr>
<tr>
<td>$I_{\text{tor}}$</td>
<td>Torsional moment of inertia.</td>
</tr>
<tr>
<td>$f_{\text{mk}}$</td>
<td>Characteristic bending strength.</td>
</tr>
<tr>
<td>$b, h$</td>
<td>Width and depth of beam.</td>
</tr>
</tbody>
</table>
For a particular Timber Strength Class (TSC), the following characteristic strength values are required to compute the other related characteristic values.

i. Bending Strength – \( f_{m,k} \)

ii. Mean Modulus of Elasticity in bending – \( E_{0,\text{mean}} \)

iii. Density - \( \rho_k \)

<table>
<thead>
<tr>
<th>SI No.</th>
<th>Property</th>
<th>Symbol</th>
<th>Wood Type</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Softwood (C)</td>
</tr>
<tr>
<td>1.</td>
<td>Tensile Strength parallel to grain</td>
<td>( f_{t,0,k} )</td>
<td>0.6 ( f_{m,k} )</td>
</tr>
<tr>
<td>2.</td>
<td>Tensile Strength perpendicular to grain</td>
<td>( f_{t,90,k} )</td>
<td>Minimum of {0.6 and ( 0.0015\rho_k )}</td>
</tr>
<tr>
<td>3.</td>
<td>Compressive Strength parallel to grain</td>
<td>( f_{c,0,k} )</td>
<td>5 ( (f_{m,k})^{0.45} )</td>
</tr>
<tr>
<td>4.</td>
<td>Compressive Strength perpendicular to grain</td>
<td>( f_{c,90,k} )</td>
<td>0.007( r_k )</td>
</tr>
<tr>
<td>5.</td>
<td>Shear Strength</td>
<td>( f_{v,k} )</td>
<td>Minimum of {3.8 and ( 0.2f_{m,k}^{0.8} )}</td>
</tr>
<tr>
<td>6.</td>
<td>Modulus of Elasticity parallel to grain</td>
<td>( E_{0,05} )</td>
<td>0.67( E_{0,\text{mean}} )</td>
</tr>
<tr>
<td>7.</td>
<td>Mean Modulus of Elasticity perpendicular to grain</td>
<td>( E_{90,\text{mean}} )</td>
<td>( E_{0,\text{mean}}/30 )</td>
</tr>
<tr>
<td>8.</td>
<td>Mean Shear Modulus</td>
<td>( G_{\text{mean}} )</td>
<td>( E_{0,\text{mean}}/16 )</td>
</tr>
<tr>
<td>9.</td>
<td>Shear Modulus</td>
<td>( G_{0,05} )</td>
<td>( E_{0,05}/16 )</td>
</tr>
</tbody>
</table>

The values of the characteristic strengths computed using the above equations, may differ with the tabulated values in Table-1 of EN 338:2003. However, in all such cases, the values obtained from the provided equations are treated as actual and is used by the program, as the values of Table-1 are based on these equations.

**Design values of Characteristic Strength**

As per clause 2.4.1, Design values of a strength property shall be calculated as:

\[
X_d = K \text{mod} \left(\frac{X_k}{Y_m}\right)
\]

Where:
X_d is design value of strength property
X_k characteristic value of strength property
γ_m is partial factor for material properties.

The member resistance in timber structure is calculated in STAAD according to the procedures outlined in EC5. This depends on several factors such as cross sectional properties, different load and material factors, timber strength class, load duration class, service class and so on. The methodology adopted in STAAD for calculating the member resistance is explained here.

**Check for Tension stresses**

If the direction of applied axial tension is parallel to the direction of timber grain alignment, the following formula should be checked per Equation 6.1 of EC-5 2004:

\[
\frac{S_{t0d}}{F_{t0d}} \leq \gamma \text{m}_{t0d}
\]

If the direction of applied axial tension is perpendicular to the direction of timber grain alignment, the following formula should be checked:

\[
\frac{S_{t90d}}{F_{t90d}} \leq \gamma \text{m}_{t90d}
\]

**Check for Compression stresses**

If the direction of applied axial compression is parallel to the direction of timber grain alignment, the following formula should be checked per Equation 6.2 of EC-5 2004:

\[
\frac{S_{c0d}}{F_{c0d}} \leq \gamma \text{m}_{c0d}
\]

If the direction of applied axial compression is perpendicular to the direction of timber grain alignment, the following formula should be checked per Equation 6.3 of EC-5 2004:

\[
\frac{S_{c90d}}{F_{c90d} \cdot K_c} \leq \gamma \text{m}_{c90d}
\]

**Check for Bending stresses**

If members are under bending stresses, the following conditions should be satisfied per Equations 6.11 and 6.12 of EC-5 2004.

**Note:** In STAAD z-z axis is the strong axis.

\[
\left( \frac{S_{mxz}}{F_{mxz}} \right) + \gamma \text{m} \left( \frac{S_{myy}}{F_{myy}} \right) \leq \gamma \text{m} \left( \frac{S_{mxz}}{F_{mxz}} \right) + \left( \frac{S_{myy}}{F_{myy}} \right) \leq K_m \left( \frac{S_{mxz}}{F_{mxz}} \right) + \left( \frac{S_{myy}}{F_{myy}} \right)
\]

**Check for Shear stresses**

Horizontal stresses are calculated and checked against allowable values per Equation 6.13 of EC-5 2004:

\[
\frac{S_{vd}}{F_{vd}} \leq \gamma \text{m}_{vd}
\]

**Check for Torsional stresses**

Members subjected to torsional stress should satisfy Equation 6.14 of EC-5 2004:

\[
\frac{S_{tor\_d}}{F_{tor\_d}} \leq \gamma \text{m}_{tor\_d}
\]
Check for combined Bending and Axial tension

Members subjected to combined action of bending and axial tension stress should satisfy Equations 6.17 and 6.18 of EC-5 2004:

Note: In STAAD z-z axis is the strong axis.

\[
\begin{align*}
\left(\frac{S_{t0d}}{F_{t0d}}\right) + \left(\frac{S_{mzd}}{F_{mzd}}\right) + K_m \left(\frac{S_{myd}}{F_{myd}}\right) &\leq \\
\left(\frac{S_{t0d}}{F_{t0d}}\right) + K_m \left(\frac{S_{mzd}}{F_{mzd}}\right) + \left(\frac{S_{myd}}{F_{myd}}\right) &\leq
\end{align*}
\]

Check for combined Bending and axial Compression

If members are subjected to bending and axial compression stress, Equations 6.19 and 6.20 of EC-5 2004 should be satisfied:

Note: In STAAD z-z axis is the strong axis.

\[
\begin{align*}
\left(\frac{S_{c0d}}{F_{c0d}}\right)^2 + \left(\frac{S_{mzd}}{F_{mzd}}\right) + K_m \left(\frac{S_{myd}}{F_{myd}}\right) &\leq \\
\left(\frac{S_{c0d}}{F_{c0d}}\right)^2 + K_m \left(\frac{S_{mzd}}{F_{mzd}}\right) + \left(\frac{S_{myd}}{F_{myd}}\right) &\leq
\end{align*}
\]

Stability check

A. Column Stability check

The relative slenderness ratios should be calculated per Equations 6.21 and 6.22 of EC-5 2004.

Note: In STAAD z-z axis is the strong axis.

\[
\begin{align*}
\lambda_{rel,z} &= \lambda_z / (\pi \cdot \frac{S_{c0k}}{E_{0.05}})^{1/2} \\
\lambda_{rel,y} &= \lambda_y / (\pi \cdot \frac{S_{c0k}}{E_{0.05}})^{1/2}
\end{align*}
\]

If both \(\lambda_{rel,z}\) and \(\lambda_{rel,y}\) are less than or equal to 0.3 the following conditions should be satisfied:

\[
\begin{align*}
\left(\frac{S_{c0d}}{F_{c0d}}\right)^2 + \left(\frac{S_{mzd}}{F_{mzd}}\right) + K_m \left(\frac{S_{myd}}{F_{myd}}\right) &\leq \\
\left(\frac{S_{c0d}}{F_{c0d}}\right)^2 + K_m \left(\frac{S_{mzd}}{F_{mzd}}\right) + \left(\frac{S_{myd}}{F_{myd}}\right) &\leq
\end{align*}
\]

In other cases, the conditions in Equations 6.23 and 6.24 of EC-5 2004 should be satisfied.

Note: In STAAD z-z axis is the strong axis.

\[
\begin{align*}
S_{c0d}/(K_{cz} F_{c0d}) + \left(\frac{S_{mzd}}{F_{mzd}}\right) + K_m \left(\frac{S_{myd}}{F_{myd}}\right) &\leq \\
S_{c0d}/(K_{cy} F_{c0d}) + K_m \left(\frac{S_{mzd}}{F_{mzd}}\right) + \left(\frac{S_{myd}}{F_{myd}}\right) &\leq
\end{align*}
\]

Where (Equations 6.25 through 6.28 of EC-5 2004):

\[
\begin{align*}
K_{cz} &= 1/|K_z + [(K_z)^2 \cdot (\lambda_{rel,z})^2]|^{1/2} \\
K_{cy} &= 1/|K_y + [(K_y)^2 \cdot (\lambda_{rel,y})^2]|^{1/2} \\
K_z &= 0.5 \left[1 + \beta_c (\lambda_{rel,z} \cdot 0.3 + (\lambda_{rel,z})^2)\right] \\
K_y &= 0.5 \left[1 + \beta_c (\lambda_{rel,y} \cdot 0.3 + (\lambda_{rel,y})^2)\right]
\end{align*}
\]

The value of \(\beta_c\) incorporated in the software is the one for solid timber (i.e., 0.2).

B. Beam Stability check
If members are subjected to only a moment about the strong axis z, the stresses should satisfy Equation 6.33 of EC-5 2004:

\[ S_{mzd}/(K_{crit} \cdot F_{mzd}) \leq \]

Where a combination of moment about the strong z-axis and compressive force exists, the stresses should satisfy Equation 6.35 of EC-5 2004 (ref. to Equations 6.32 and 6.34 of the same):

\[ (S_{mzd}/(K_{crit} \cdot F_{mzd}))^2 + S_{c0d}/(K_{cz} \cdot F_{c0d}) \leq \]

Where:

\[ K_{crit} = 1.0 \text{ when } \lambda_{rel,m} \leq 0.75 \]
\[ K_{crit} = 1.56 - 0.75 \cdot \lambda_{rel,m} \text{ when } 0.75 < \lambda_{rel,m} \leq 1.4 \]
\[ K_{crit} = 1/(\lambda_{rel,m})^2 \text{ when } 1.4 < \lambda_{rel,m} \]
\[ \lambda_{rel,m} = (f_{mk}/S_{m,crit})^{1/2} \]

For hardwood, use Equation 6.30 of EC-5 2004:

\[ S_{m,crit} = \pi \cdot (E_{0.05} \cdot I_y \cdot G_{0.05} \cdot I_{tor})^{1/2}/(l_{ef} \cdot W_z) \]

For softwood, use Equation 6.31 of EC-5 2004:

\[ S_{m,crit} = 0.78 \cdot b^2 \cdot E_{0.05}/(h \cdot l_{ef}) \]

**D5.E.3 Design Parameters**

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Depending on the model being designed, the user may have to change some or all of the parameter default values. Some parameters are unit dependent and when altered, the new setting must be compatible with the active “unit” specification.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

**Table 160: Timber Design EC 5: Part 1-1 Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as TIMBER EC5 Design Code to follow. See section TR.52.1 Timber Design Parameter Specifications (on page 2857).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| ALPHA          | 0.0           | Angle of inclination of load to the grain alignment. (Ref. Cl.6.1.1, Cl. 6.1.2, Cl.6.1.3, Cl.6.1.4)  
                      0.0 = Load parallel to grain  
                      90.0 = Load Perpendicular to grain |
| DFF            | None          | "Deflection Length" / Max. Allowable Net Final Local Deflection.  
                      In this case, deflection check will be performed, if both the parameters SERV and DFF are present with specific values. For appropriate range of values, please refer Cl.7.2 (Table 7.2) |
| DJ1            |               | Start node number for a physical member under consideration for Deflection Check. |
| DJ2            |               | End node number for a physical member under consideration for Deflection Check. |
| KC90           | 1.0           | Factor taking into account the load configuration, possibility of splitting and degree of compressive deformation. (Ref.Cl.6.1.5-(2))  
                        • Range: 1.0 ≤ KC90 ≤ 4.0  
                        • Other than the default value, user may specify any value within the range, depending on load-position, load-dispersion, contact length at support locations etc. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>KLEF</td>
<td>1.0</td>
<td>Effective Length Factor to check Lateral Torsional Buckling (Ref. Table 6.1). Factor multiplied by the span of the beam and depends on the support conditions and load configurations. The user will put the appropriate value from the Table 6.1. Required only for MTYP value of 1 (Beam).</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective Length Factor for Local-y-axis. (Ref. Cl.6.3.2), for the computation of the relative slenderness ratios.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective Length Factor for Local-z-axis. (Ref. Cl.6.3.2), for the computation of the relative slenderness ratios.</td>
</tr>
<tr>
<td>LDC</td>
<td>1</td>
<td>Load Duration Class (Ref. Cl. 2.3.1.2), required to get the K-MOD value from Table – 3.1.</td>
</tr>
<tr>
<td>MTYP</td>
<td>0</td>
<td>Member Type: Beam/Column. (Ref. Cl.6.3.2, Cl.6.3.3)</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual to allowable value.</td>
</tr>
</tbody>
</table>
## D5.E.4 Verification Examples

### D5.E.4.1 Verification Example No. 1 - Timber Column

A Timber Column of length 1.0 meter, having c/s dimension of 73 mm X 198 mm, is subjected to an axial compressive force of 50.0 kN. Design the member for the ultimate limit state.

Material properties:

- Timber class: C24
- Service classes: Class 2, moisture content ≤ 20%
- Load duration classes: Medium-term

---

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SCL</td>
<td>3</td>
<td>Service Class (Ref. Cl.2.3.1.3)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = Class 1, Moisture content ≤ 12%</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.0 = Class 2, Moisture content ≤ 20%</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3.0 = Class 3, Moisture content &gt; 20%</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>Degree/Level of Details of design output results.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = Print the design output at the minimal detail level</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.0 = Print the design output at the intermediate detail level</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3.0 = Print the design output that the maximum detail level</td>
</tr>
<tr>
<td>TSC</td>
<td>6 (C24)</td>
<td>Timber Strength Class (Ref. Reference EN338 – 2003)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Softwood: 1 = C14, 2 = C16, 3 = C18, 4 = C20, 5 = C22, 6 = C24, 7 = C27, 8 = C30, 9 = C35, 10 = C40, 11 = C45, 12 = C50.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Hardwood: 13 = D30, 14 = D35, 15 = D40, 16 = D50, 17 = D60, 18 = D70.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This TSC definition will calculate the corresponding characteristic strength values using the equations as given in BS-EN-338, Annex - A.</td>
</tr>
</tbody>
</table>
Cross section properties:

Length of the member is 1 m.
Rectangular cross section, b = 73 mm, h = 198 mm,
Effective cross sectional area A = 14,454 mm²,
Radius of gyration of cross section about y-axis \( r_y = 21 \text{ mm} \),
Radius of gyration of cross section about z-axis \( r_z = 57 \text{ mm} \),
Section modulus of cross section about z-axis: \( W_z = 4.770 \times 10^5 \text{ mm}^3 \)
Section modulus of cross section about y-axis: \( W_y = 1.759 \times 10^5 \text{ mm}^3 \)

Solution

Characteristic material properties for timber:
Modification factor \( K_{mod} = 0.80 \) ...from table 3.1
Material factors \( \gamma_m = 1.30 \) ... from table 2.3
\[
f_{c0k} = 21.00 \text{ N/mm}^2
\]
\[
F_{c0d} = (K_{mod} f_{c0k})/\gamma_m = (0.80 \times 21.00)/1.30 = 12.92 \text{ N/mm}^2 \quad [\text{Cl. 2.4.1(1)P}]
\]

Cross section loads:
\( F_x = 50.000 \text{ kN} \)

Compression parallel to the grain:
\[
S_{c0d} = (1000 \times F_x)/A = (1000 \times 50.000)/14454 = 3.46 \text{N/mm}^2 < 12.92 \text{N/mm}^2 \quad (F_{c0d})
\]

The ratio of actual compressive stress to allowable compressive strength:
\[
S_{c0d}/F_{c0d} = 3.46 / 12.92 = 0.268 < 1.0 \quad [\text{Cl. 6.1.4.(1)P}]
\]

Check for Slenderness:

Slenderness ratios:
\[
\lambda_z = (1000/57) = 17.54
\]
\[
\lambda_y = (1000/21) = 47.62
\]
\[
E_{0,\text{mean}} = 1.1031 \text{ kN/m}^2
\]

As timber grade is C24 (i.e., Soft Wood)
\[
E_{0,05} = 0.67 \times E_{0,\text{mean}} = 0.739 \text{ kN/m}^2
\]

[Annex A,EN 338:2003]
\[
\lambda_{rel,z} = \lambda_z/\pi \left( f_{c0k}/E_{0,05}\right)^{1/2} = 17.54/\pi(21.00/0.739)^{1/2} = 0.298
\]
\[
\lambda_{rel,y} = \lambda_y/\pi \left( f_{c0k}/E_{0,05}\right)^{1/2} = 47.62/\pi(21.00/0.739)^{1/2} = 0.809
\]

Since, \( \lambda_{rel,y} \) is greater than 0.3, following conditions should be satisfied:
\[
S_{c0d}/(K_z F_{c0d}) + (S_{mzd}/F_{mzd}) + K_m (S_{myd}/F_{myd}) \leq
\]
\[
S_{c0d}/(K_y F_{c0d}) + K_m (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \leq
\]

Where:
\[
K_z = 0.5 \left[ 1 + \beta_c (\lambda_{rel,z} \cdot 0.3) + (\lambda_{rel,z})^2 \right] = 0.5 \left[ 1 + 0.2(0.298 \cdot 0.3) + (0.298)^2 \right] = 0.541
\]
\[
K_y = 0.5 \left[ 1 + \beta_c (\lambda_{rel,y} \cdot 0.3) + (\lambda_{rel,y})^2 \right] = 0.5 \left[ 1 + 0.2(0.809 \cdot 0.3) + (0.809)^2 \right] = 0.878
\]
Kcz = 1/(Kz + [(Kz)^2 - (λ_{rel,z})^2]^{1/2}) = 1/(0.541 + [(0.541)^2 - (0.298)^2]^{1/2}) = 1.008

Kcy = 1/(Ky + [(Ky)^2 - (λ_{rel,y})^2]^{1/2}) = 1/(0.878 + [(0.878)^2 - (0.809)^2]^{1/2}) = 0.820

For Rectangular cross section Km = 0.70. The member is subjected to Compression only, so actual bending stress is zero.

\[ S_{c0d}/(Kcz F_{c0d}) + (S_{myd}/F_{myd}) + Km (S_{myd}/F_{myd}) = 3.46/(1.008 \cdot 12.92) + 0.0 + 0.0 = 0.266 \]

\[ S_{c0d}/(Kcy F_{c0d}) + Km (S_{myd}/F_{myd}) + (S_{myd}/F_{myd}) = 3.46 / (0.820 \cdot 12.92) + 0.0 + 0.0 = 0.326 \]

Hence the critical ratio is 0.326 < 1.0 and the section is safe.

Comparison

Table 161: EC 5: Part 1-1 Verification Example 1

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio (Cl.6.3.2)</td>
<td>0.326</td>
<td>0.327</td>
<td>none</td>
</tr>
</tbody>
</table>

Input File

The following file is included as
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Eur\EC5 ver 1.std.

STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 1.0 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC WOOD
E 1.10316e+007
POISSON 0.15
DENSITY 0.00231749
ALPHA 5.5e-006
END DEFINE MATERIAL
CONSTANTS
MATERIAL WOOD MEMB 1
MEMBER PROPERTY
1 PRIS YD 0.198 ZD 0.073
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
JOINT LOAD
2 FX -50
PERFORM ANALYSIS
PARAMETER
CODE TIMBER EC5
ALPHA 0 ALL
LDC 3 ALL
SCL 2 ALL
TSC 6 ALL
**Output**

The member checking part of the output file:

```
STAAD.Pro CODE CHECKING - (EC5 )

******************************

ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)

MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
  FX MY MZ LOCATION

=======================================================================

1   PRIS ZD = 0.073 YD = 0.198 
     PASS CL.6.3.2 0.327         1
     50.00 C 0.00 0.00 0.0000

| AX = 0.01  IY = 0.00  IZ = 0.00 |
| LEZ = 1.00  LEY = 1.00 |

ALLOWABLE STRESSES: (NEW MMS)
  FBY = 14.769  FBZ = 14.769
  FC = 12.859

ACTUAL STRESSES : (NEW MMS)
  fby = 0.000  fbz = 0.000
  fc = 3.459
```

---

**D5.E.4.2 Verification Example No. 2**

A Timber Column of length 1.0 meter, having c/s dimension of 73 mm X 198 mm, is subjected to an axial compressive force of 5.0 kN and moments of 2.0 kN.m and 1.0 kN.m about its major and minor axes respectively. Design the member for the ultimate limit state.

Material properties:

- Timber Strength Class: C24
- Service classes: Class 2, moisture content <=20%
- Load duration: Medium-term

Cross section properties:

- Length of the member is 1 m.
- Rectangular cross section, b = 73 mm, h = 198 mm,
- Effective cross sectional area A = 14454 mm²,
Radius of gyration of cross section about y-axis \( r_y = 21 \, \text{mm} \),
Radius of gyration of cross section about z-axis \( r_z = 57 \, \text{mm} \),
Section modulus of cross section about z-axis: \( W_z = 4.770 \times 10^5 \, \text{mm}^3 \)
Section modulus of cross section about y-axis: \( W_y = 1.759 \times 10^5 \, \text{mm}^3 \)

Solution

Characteristic material properties for timber:
Modification factor \( K_{mod} = 0.80 \) ...from table 3.1
Material factors \( \gamma_m = 1.30 \) ... from table 2.3
\( f_{c0k} = 21.00 \, \text{N/mm}^2 \)
\( E_0,05 = 7370 \, \text{N/mm}^2 \)
\( f_{myk} = 24.00 \, \text{N/mm}^2 \)

\( f_{mzk} = 24.00 \, \text{N/mm}^2 \)

\( F_{c0d} = (K_{mod} \cdot f_{c0k})/\gamma_m = (0.80 \cdot 21.00)/1.30 = 12.92 \, \text{N/mm}^2 \) \[Cl 2.4.1(1)P\]
\( f_{myd} = K_{mod} \cdot f_{myk}/\gamma_m = (0.80 \cdot 24.00)/1.30 = 14.77 \, \text{N/mm}^2 \)

Cross section loads:
\( F_x = 5.000 \, \text{kN} \)
\( M_z = 2.000 \, \text{kN} \cdot \text{m} \)
\( M_y = 1.000 \, \text{kN} \cdot \text{m} \)

Check for Slenderness:

Slenderness ratios:
\( \lambda_z = (1000/57) = 17.54 \)
\( \lambda_y = (1000/21) = 47.62 \)
\( \lambda_{rel,z} = \frac{\lambda_z}{\pi \cdot \sqrt{\left(\frac{f_{c0k}}{E_0,05}\right)^2}} = 17.54/\pi \cdot \sqrt{(21.00/7370)} \cdot 1/2 = 0.298 \)
\( \lambda_{rel,y} = \frac{\lambda_y}{\pi \cdot \sqrt{\left(\frac{f_{c0k}}{E_0,05}\right)^2}} = 47.62/\pi \cdot \sqrt{(21.00/7370)} \cdot 1/2 = 0.809 \)

Since, \( \lambda_{rel,y} \) is greater than 0.3, following conditions should be satisfied [Cl 6.3.2.3]:
\( S_{c0d}/(K_{cz} \cdot F_{c0d}) + (S_{mzd}/F_{mzd}) + K_m \cdot (S_{myd}/F_{myd}) \leq \)
\( S_{cz0d}/(K_{cy} \cdot F_{c0d}) + K_m \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \leq \)

Where:
\( K_{cz} = 0.5 \cdot [1 + 0.3 \cdot (\lambda_{rel,z} \cdot 0.3) + (\lambda_{rel,z})^2] = 0.50 \cdot [1 + 0.2 \cdot (0.298 \cdot 0.3) + (0.298)^2] = 0.541 \)
\( K_y = 0.5 \cdot [1 + 0.3 \cdot (\lambda_{rel,y} \cdot 0.3) + (\lambda_{rel,y})^2] = 0.50 \cdot [1 + 0.2 \cdot (0.809 \cdot 0.3) + (0.809)^2] = 0.878 \)
\( K_{cz} = 1/(K_{cz} + (K_{cz})^2 \cdot (\lambda_{rel,z})^2) = 1/(0.541 + (0.541)^2 \cdot (0.298)^2) \cdot 1/2 = 1.008 \)
\( K_{cy} = 1/(K_{cy} + (K_{cy})^2 \cdot (\lambda_{rel,y})^2) = 1/(0.878 + (0.878)^2 \cdot (0.809)^2) \cdot 1/2 = 0.820 \)

For Rectangular cross section \( K_m = 0.70 \).
\( S_{c0d} = (1000 \cdot F_x/A) = (1000 \cdot 5.000)/14454 = 0.35 \, \text{N/mm}^2 \)
\[ S_{max} = \frac{(10^6 \cdot M_z)}{W_z} = \frac{(10^6 \cdot 2.000)}{(4.770 \times 10^5)} = 4.19 \text{ N/mm}^2 \]

\[ S_{myd} = \frac{(10^6 \cdot M_y)}{W_y} = \frac{(10^6 \cdot 1.000)}{(1.759 \times 10^5)} = 5.69 \text{ N/mm}^2 \]

Combined stress ratio:

\[ S_{c0d}/(Kcz \cdot F_{c0d}) + \frac{S_{mzd}}{F_{mzd}} + Km \cdot \frac{S_{myd}}{F_{myd}} = \frac{0.35}{(1.008 \cdot 12.92)} + 4.19/14.77 + 0.70(5.69/14.77) \]

\[ = 0.266 \]

Hence the critical ratio is 0.616 < 1.0 and the section is safe.

Comparison

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio (Cl. 6.3.2)</td>
<td>0.616</td>
<td>0.616</td>
<td>none</td>
</tr>
</tbody>
</table>

Input File

The following file is included as
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Eur\EC5 ver 2.std.

STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1 0;
MEMBER INCIDENCES 1 1 2;
DEFINE MATERIAL START
ISOTROPIC WOOD
E 1.10316e+007
POISSON 0.15
DENSITY 0.00231749
ALPHA 5.5e-006
END DEFINE MATERIAL
CONSTANTS
MATERIAL WOOD MEMB 1
MEMBER PROPERTY
1 PRIS YD 0.198 ZD 0.073
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
JOINT LOAD
2 FY -5.0 MX 1.0 MZ 2.0
PERFORM ANALYSIS
PARAMETER
CODE TIMBER EC5
ALPHA 0 ALL
LDC 3 ALL
SCL 2 ALL
TSC 6 ALL
TRACK 2 ALL
Output

The member checking part of the output file:

```
STAAD.Pro CODE CHECKING - (EC5)
***************

ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/CRITICAL COND/RATIO/LOADING/
FX            MY             MZ       LOCATION
=======================================================================
1    PRIS ZD =      0.073 YD =      0.198
      PASS      CL.6.3.2            0.616         1
      5.00 C          1.00          -2.00     0.0000
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>AX =     0.01  IY =           0.00  IZ =           0.00</td>
</tr>
<tr>
<td>LEZ =     1.00  LEY =     1.00</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>ALLOWABLE STRESSES: (NEW MMS)</td>
</tr>
<tr>
<td>FBY  =       14.769 FBZ   =       14.769</td>
</tr>
<tr>
<td>FC   =       12.859</td>
</tr>
<tr>
<td>ACTUAL STRESSES : (NEW MMS)</td>
</tr>
<tr>
<td>fby  =        5.686 fbz   =        4.193</td>
</tr>
<tr>
<td>fc   =        0.346</td>
</tr>
<tr>
<td>--------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
```

D6. French Codes

D6.A. French Codes - Steel Design per CM66-1977 (French)

STAAD.Pro is capable of performing steel design based on the French code CM66, 1977 edition Centre Technique Industriel de la Construction Metallique (Industrial Technical Center of Metal Construction) publication entitled Design Rules for Structural Steelwork.

D6.A.1 General Comments

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned according to the limit states of which they would become unfit for their intended use. Two major categories of limit-states are recognized: ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability; that in serviceability is deflection. Appropriate load and resistance factors are used so that uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.
In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria, as augmented by the designer in specification of allowable member depths, desired section type, or other related parameters. The code checking portion of the program verifies that code requirements for each selected section are met and also identifies the governing criteria.

The next few sections describe the salient features of STAAD.Pro implementation of “Design Rules for Structural Steelwork.” A detailed description of the design process, along with its underlying concepts and assumptions, is available in the specification document.

D6.A.2 Basis of Methodology

The Design Rules for Structural Steelwork (Revision 80) permits the usage of elastic analysis. Thus, in STAAD.Pro, linear elastic analysis method is used to obtain the forces and moments in the members. However, strength and stability considerations are based on the principles of plastic behavior. Axial compression buckling and lateral torsional buckling are taken into consideration for calculation of axial compression resistance and flexural resistance of members. Slenderness calculations are made and overall geometric stability is checked for all members.

D6.A.3 Member Capacities

The member strengths are calculated in STAAD.Pro according to the procedures outlined in section 4 of this specification. Note that the program automatically considers co-existence of axial force, shear and bending in calculating section capacities.

For axial tension capacity, procedures of section 4.2 are followed. For axial compression capacity, formulas of section 5.3 are used.

Moment capacities about both axes are calculated using the procedures of sections 4.5 and 4.6. Lateral torsional buckling is considered in calculating ultimate twisting moment per section 5.22 of the specification. The parameter UNL (see D6.A.5 Design Parameters (on page 1873)) must be used to specify the unsupported length of the compression flange for a laterally unsupported member. Note that this length is also referred to as twisting length.

D6.A.4 Combined Axial Force and Bending

The procedures of sections 4.55 and 5.32 are implemented for interaction of axial forces and bending. Appropriate interaction equations are used and the governing criterion is determined.

D6.A.5 Design Parameters

The design parameters outlined in the following table may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program, thus allowing the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected as frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.
### Table 163: French Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>FRENCH</td>
<td>Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>0.0</td>
<td>0.0 = design only for end moments and those at locations specified by SECTION command. 1.0 = calculate moments at tenth points long the beam, and use maximum Mz for design.</td>
</tr>
<tr>
<td>C1</td>
<td>1.0</td>
<td>Parameter used in clause 5.21 in the calculation of M(D), the critical twisting moment and as shown in CM 66 Addendum 80, table 5, usual range from 0.71 to 4.10</td>
</tr>
<tr>
<td>C2</td>
<td>1.0</td>
<td>Parameter used in clause 5.21 in the calculation of M(D), the critical twisting moment and as shown in CM 66 Addendum 80, table 5, usual range from 0.0 to 1.56</td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>“Deflection Length” divided by the Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot; (See Note 1)</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot; (See Note 1)</td>
</tr>
<tr>
<td>DMAX</td>
<td>100.0 cm.</td>
<td>Maximum allowable depth (used in member selection).</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 cm.</td>
<td>Minimum allowable depth (used in member selection).</td>
</tr>
<tr>
<td>FYLD</td>
<td>250.0 MPa</td>
<td>Yield strength of steel.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value for axial compression buckling about local Y-axis. Usually, this is the minor axis.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value for axial compression buckling about local Z-axis. Usually, this is the major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio about Y-axis for axial compression.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Length to calculate slenderness ratio about Z-axis for axial compression.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual load effect and design strength.</td>
</tr>
<tr>
<td>SAME</td>
<td>0.0</td>
<td>Controls the sections to try during a SELECT process.</td>
</tr>
</tbody>
</table>
| TRACK          | 0.0           | 0.0 = Suppress printing of all design strengths.  
|                |               | 1.0 = Print all design strengths. |
| UNF            | 1.0           | Same as above provided as a fraction of member length. |
| UNL            | Member Length | Unsupported length of compression flange for calculating moment resistance. |

Note a: For angles, if the original section is an equal angle, then the selected section will be an equal angle and vice versa for unequal angles.
D6.A.6 Code Checking and Member Selection

Both code checking and member selection options are available in the STAAD.Pro implementation of CM 66 (Revn. 80).

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

D6.A.7 Tabulated Results of Steel Design

Results of code checking and member selection are presented in the output file in a tabular format.

Note: COND CRITIQUE refers to the section of the CM 66 (Revn. 80) specification which governed the design.

If the TRACK parameter is set to 1.0, calculated member capacities will be printed. The following is a detailed description of printed items:

- **PC**: Member Compression Capacity
- **TR**: Member Tension Capacity
- **MUZ**: Member Moment Capacity (about z-axis)
- **MUY**: Member Moment Capacity (about y-axis)
- **VPZ**: Member Shear Capacity (z-axis)
- **VPY**: Member Shear Capacity (y-axis)

STAAD.Pro contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.
- Specify design parameter values, if different from the default values.

These operations may be repeated by the user any number of times depending upon the design requirements.

STAAD.Pro supports steel design of wide flange, S, M, HP shapes, angle, double angle, channel, double channel, beams with cover plate, composite beams and code checking of prismatic properties.

Sample Input data for Steel Design:

```
UNIT  METER
PARAMETER
CODE  FRENCH
NSF  0.85 ALL
UNL  10.0 MEMBER 7
KY   1.2 MEMBER 3 4
RATIO 0.9 ALL
TRACK 1.0 ALL
CHECK CODE ALL
```
D6.A.8 Built-in French Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered for these members.

An example of the member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Following are the descriptions of different types of sections.

D6.A.8.1 IPE Shapes

These shapes are designated in the following way.

<table>
<thead>
<tr>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 15 TA ST IPE140</td>
</tr>
<tr>
<td>20 TO 30 TA ST IPEA120</td>
</tr>
<tr>
<td>33 36 TO 46 BY 2 TA ST IPER180</td>
</tr>
</tbody>
</table>

D6.A.8.2 HE shapes

HE shapes are specified as follows.

<table>
<thead>
<tr>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 5 TA ST HEA120A</td>
</tr>
<tr>
<td>7 10 TA ST HEM140</td>
</tr>
<tr>
<td>13 14 TA ST HEB100</td>
</tr>
</tbody>
</table>

D6.A.8.3 IPN Shapes

The designation for the IPN shapes is similar to that for the IPE shapes.

<table>
<thead>
<tr>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 TO 35 TA ST IPN200</td>
</tr>
<tr>
<td>23 56 TA ST IPN380</td>
</tr>
</tbody>
</table>

D6.A.8.4 T Shapes

Tee sections are not input by their actual designations, but instead by referring to the I beam shapes from which they are cut. For example,

<table>
<thead>
<tr>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 5 TA T IPE140</td>
</tr>
<tr>
<td>2 8 TA T HEM120</td>
</tr>
</tbody>
</table>

D6.A.8.5 U Channels

Shown below is the syntax for assigning 4 different names of channel sections.

<table>
<thead>
<tr>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 5 TA ST UAP100</td>
</tr>
<tr>
<td>6 TO 10 TA ST UPN220</td>
</tr>
<tr>
<td>11 TO 15 TA ST UPN240A</td>
</tr>
<tr>
<td>16 TO 20 TA ST UAP250A</td>
</tr>
</tbody>
</table>
D6.A.8.6 Double U Channels

Back to back double channels, with or without a spacing between them, are available. The letter D in front of the section name will specify a double channel.

<table>
<thead>
<tr>
<th>Member</th>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>TA D UAP150</td>
<td>back-to-back double channel UAP150 with no spacing</td>
</tr>
<tr>
<td>17</td>
<td>TA D UAP250A SP 0.5</td>
<td>double channel UAP250A with spacing of 0.5 length units</td>
</tr>
</tbody>
</table>

In the above set of commands, member 11 is a back-to-back double channel UAP150 with no spacing in between. Member 17 is a double channel UAP250A with a spacing of 0.5 length units between the channels.

D6.A.8.7 Angles

Two types of specification may be used to describe an angle. The standard angle section is specified as follows:

<table>
<thead>
<tr>
<th>Member</th>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>20 TA ST L30X30X2.7</td>
<td>angle with legs of length 30mm and a leg thickness of 2.7mm</td>
</tr>
</tbody>
</table>

The above section signifies an angle with legs of length 30mm and a leg thickness of 2.7mm. This specification may be used when the local Z axis corresponds to the z-z axis specified in Chapter 2. If the local Y axis corresponds to the z-z axis, type specification "RA" (reverse angle) should be used instead of ST.

<table>
<thead>
<tr>
<th>Member</th>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>21 TA RA L25X25X4</td>
<td>reverse angle with legs of length 25mm and a leg thickness of 4mm</td>
</tr>
<tr>
<td>22</td>
<td>24 TA RA L100X100X6.5</td>
<td>reverse angle with legs of length 100mm and a leg thickness of 6.5mm</td>
</tr>
</tbody>
</table>

Note that if the leg thickness is a round number such as 4.0, only the number 4 appears in the section name, the decimal part is not part of the section name.

D6.A.8.8 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by means of input of the words SD or LD, respectively, in front of the angle size. In case of an equal angle, either SD or LD will serve the purpose.

<table>
<thead>
<tr>
<th>Member</th>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>33</td>
<td>35 TA SD L30X20X4 SP 0.6</td>
<td>short leg back-to-back double angle with legs of length 30mm and 20mm and a leg thickness of 4mm</td>
</tr>
<tr>
<td>37</td>
<td>39 TA LD L80X40X6</td>
<td>long leg back-to-back double angle with legs of length 80mm and 40mm and a leg thickness of 6mm</td>
</tr>
<tr>
<td>43</td>
<td>47 TA LD L80X80X6.5 SP 0.75</td>
<td>long leg back-to-back double angle with legs of length 80mm and a leg thickness of 6.5mm</td>
</tr>
</tbody>
</table>

D6.A.8.9 Tubes (Rectangular or Square Hollow Sections)

Section names of tubes, just like angles, consist of the depth, width and wall thickness as shown below.

<table>
<thead>
<tr>
<th>Member</th>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>64</td>
<td>78 TA ST TUB50252.7</td>
<td>tube with a depth of 50mm, width of 25mm and a wall thickness of 2.7mm</td>
</tr>
<tr>
<td>66</td>
<td>73 TA ST TUB2001008.0</td>
<td>tube with a depth of 200mm, width of 100mm and a wall thickness of 8.0mm</td>
</tr>
</tbody>
</table>

Members 64 and 78 are tubes with a depth of 50mm, width of 25mm and a wall thickness of 2.7mm. Members 66 and 73 are tubes with a depth of 200mm, width of 100mm and a wall thickness of 8.0mm. Unlike angles, the "0" in the thickness is part of the section name.

Tubes can also be input by their dimensions instead of by their table designations. For example,

<table>
<thead>
<tr>
<th>Member</th>
<th>Commands</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>TA ST TUBE DT 8.0 WT 6.0 TH 0.5</td>
<td>tube that has a depth of 8 length units, width of 6 length units, and a wall thickness of 0.5 length units</td>
</tr>
</tbody>
</table>

is a tube that has a depth of 8 length units, width of 6 length units, and a wall thickness of 0.5 length units. Only code checking, no member selection, will be performed for TUBE sections specified in this way.
D6.A.8.10 Pipes (Circular Hollow Sections)

To designate circular hollow sections, use PIP followed by numerical value of the diameter and thickness of the section in mm omitting the decimal portion of the value provided for the diameter. The following example illustrates the designation.

<p>| | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>28</td>
<td>TA</td>
<td>ST</td>
<td>PIP422.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>64</td>
<td>78</td>
<td>TA</td>
<td>ST</td>
<td>PIP21912.5</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Members 8 to 28 are pipes 42.4mm in dia, having a wall thickness of 2.6mm. Members 3, 64 and 78 are pipes 219.1mm in dia, having a wall thickness of 12.5mm.

Circular hollow sections may also be provided by specifying the outside and inside diameters of the section. For example,

<p>| | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9</td>
<td>TA</td>
<td>ST</td>
<td>PIPE OD 25.0 ID 20.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

specifies a pipe with outside dia. of 25 length units and inside dia. of 20 length units. Only code checking, no member selection will be performed if this type of specification is used.

D6.A.8.11 Example

Sample file containing French shapes:

```
STAAD SPACE
UNIT METER KN
JOINT COORD
1 0 0 15 140 0 0
MEMB INCI
1 1 2 14
UNIT CM
MEMBER PROPERTIES FRENCH
* IPE SHAPES
1 TA ST IPEA120
* IPN SHAPES
2 TA ST IPN380
* HE SHAPES
3 TA ST HEA200
* T SHAPES
4 TA T HEM120
* U CHANNELS
5 TA ST UAP100
* DOUBLE U CHANNELS
6 TA D UAP150 SP 0.5
* ANGLES
7 TA ST L30X30X2.7
* REVERSE ANGLES
8 TA RA L25X25X4
* DOUBLE ANGLES - SHORT LEGS BACK
* TO BACK
9 TA SD L30X20X4 SP 0.25
* DOUBLE ANGLES - LONG LEGS BACK
* TO BACK
10 TA LD L80X40X6 SP 0.75
* TUBES (RECTANGULAR OR SQUARE
* HOLLOW SECTIONS)
11 TA ST TUB50252.7
* TUBES (RECTANGULAR OR SQUARE
* HOLLOW SECTIONS)
12 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```
D7. German Codes

D7.A. German Codes - Steel Design per DIN 18800 Code

STAAD.Pro is capable of performing concrete design based on the German code DIN 18800, Parts 1 & 2: Stahlbauten - Teil 1: Bemessung und Konstruktion (Steel structures - Part 1: Design and construction) and Stahlbauten - Teil 2: Stabilitätsfälle - Knicken von Stäben und Stabwerken (Steel structures - Part 2: Analysis of safety against buckling of linear members and frames)

D7.A.1 General

This section presents some general statements regarding the implementation of the DIN code. The design philosophy and procedural logistics are based on the principles of elastic analysis and allowable stress design. Facilities are available for member selection as well as code checking. Two major failure modes are recognized: failure by overstressing and failure by stability considerations. The following sections describe the salient features of the design approach.

Members are proportioned to resist the design loads without exceeding the allowable stresses or capacities and the most economical section is selected on the basis of the least weight criteria. The code checking part of the program also checks the slenderness requirements and the stability criteria. It is recommended that you use the following steps in performing the steel design:

1. Specify the geometry and loads and perform the analysis.
2. Specify the design parameter values if different from the default values.
3. Specify whether to perform code checking or member selection.

D7.A.2 Analysis Methodology

The elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions you provide. You are allowed complete flexibility in providing loading specifications and in using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D7.A.3 Member Property Specifications

For specification of member properties of standard German steel sections, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to G.6 Member Properties (on page 2322).
D7.A.4 Built-in German Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, these properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered for these members during the analysis. An example of member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained using the tools of the graphical user interface.

Following are the descriptions of different types of sections.

Refer to G.6.2 Built-In Steel Section Libraries (on page 2325) for additional information.

D7.A.4.1 IPE Shapes

These shapes are designated in the following way:

<table>
<thead>
<tr>
<th>Section</th>
<th>Depth</th>
<th>Flange Width</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>20 30 TA ST IPEA120</td>
<td>20 TO 30 TA ST IPEA120</td>
<td></td>
<td></td>
</tr>
<tr>
<td>33 46 BY 2 TA ST IPER140</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

D7.A.4.2 HE Shapes

The designation for HE shapes is similar to that for IPE shapes.

<table>
<thead>
<tr>
<th>Section</th>
<th>Depth</th>
<th>Flange Width</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 35 TA ST HEB300</td>
<td>25 TO 35 TA ST HEB300</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23 56 TA ST HEA160</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

D7.A.4.3 I Shapes

I shapes are identified by the depth of the section. The following example illustrates the designation.

<table>
<thead>
<tr>
<th>Section</th>
<th>Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>14 15 TA ST I200</td>
<td>14 15 TA ST I200</td>
</tr>
</tbody>
</table>

(indicates an I-section with 200mm depth)

D7.A.4.4 T Shapes

Tee sections are not input by their actual designations, but instead by referring to the I beam shapes from which they are cut. For example,

<table>
<thead>
<tr>
<th>Section</th>
<th>Type</th>
<th>Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 5 TA T HEA220</td>
<td>1 5 TA T HEA220</td>
<td></td>
</tr>
<tr>
<td>2 8 TA T IPE120</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

D7.A.4.5 U Channels

The example below provides the command for identifying two channel sections. The former (U70X40) has a depth of 70mm and a flange width of 40mm. The latter (U260) has a depth of 260mm.

<table>
<thead>
<tr>
<th>Section</th>
<th>Type</th>
<th>Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>11 TA D U70X40</td>
<td>11 TA D U70X40</td>
<td></td>
</tr>
<tr>
<td>27 TA D U260</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

STAAD.Pro 1881 User Manual
**D7.A.4.6 Double Channels**

Back-to-back double channels, with or without spacing between them, are available. The letter "D" in front of the section name will specify a double channel, e.g., D U180. The spacing between the double channels is provided following the expression "SP".

<table>
<thead>
<tr>
<th>TA</th>
<th>Section</th>
<th>Width</th>
<th>Height</th>
<th>Spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>TA D U180</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>TA D U280</td>
<td></td>
<td></td>
<td>0.5</td>
</tr>
</tbody>
</table>

(Indicates 2 channels back-to-back spaced at 0.5 length units)

**D7.A.4.7 Angles**

Two types of specifications may be used to describe an angle. The standard angle section is specified as follows:

<table>
<thead>
<tr>
<th>TA</th>
<th>Width</th>
<th>Height</th>
<th>Leg Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>20</td>
<td>20</td>
<td>2.5</td>
</tr>
</tbody>
</table>

The above section signifies an angle with legs of length 20mm and a leg thickness of 2.5mm. The above specification may be used when the local z-axis corresponds to the Z-Z axis specified in Chapter 2. If the local y-axis corresponds to the Z-Z axis, type specification "RA" (reverse angle) may be used.

<table>
<thead>
<tr>
<th>TA</th>
<th>Width</th>
<th>Height</th>
<th>Leg Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>40</td>
<td>20</td>
<td>5</td>
</tr>
</tbody>
</table>

**D7.A.4.8 Double Angles**

Short leg back-to-back or long leg back-to-back double angles can be specified by using the word SD or LD, respectively, in front of the angle size. In case of an equal angle, either SD or LD will serve the purpose. Spacing between the angles is provided by using the word SP and the spacing value following the section name.

<table>
<thead>
<tr>
<th>TA</th>
<th>Width</th>
<th>Height</th>
<th>Leg Thickness</th>
<th>Spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>40</td>
<td>20</td>
<td>4</td>
<td>0.5</td>
</tr>
<tr>
<td>21</td>
<td>40</td>
<td>20</td>
<td>4</td>
<td>0.5</td>
</tr>
</tbody>
</table>

**D7.A.4.9 Pipes (Circular Hollow Sections)**

To designate circular hollow sections, use PIP followed by numerical value of the diameter and thickness of the section in mm omitting the decimal section of the value provided for diameter. The following example will illustrate the designation.

<table>
<thead>
<tr>
<th>TA</th>
<th>Diameter</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>602.9</td>
<td></td>
</tr>
</tbody>
</table>
* (60.3mm dia, 2.9mm wall thickness)

Circular hollow sections may also be provided by specifying the outside and inside diameters of the section. For example,

<table>
<thead>
<tr>
<th>TA</th>
<th>Outside Diameter</th>
<th>Inside Diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>25.0</td>
<td>20.0</td>
</tr>
</tbody>
</table>

specifies a pipe with outside dia. of 25 and inside dia. of 20 in current length units. Only code checking and no member selection will be performed if this type of specification is used.

**D7.A.4.10 Tubes (Rectangular or Square Hollow Sections)**

Tube names are input by their dimensions. For example,

<table>
<thead>
<tr>
<th>TA</th>
<th>Width</th>
<th>Height</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
<td>25</td>
<td>60</td>
<td>3.6</td>
</tr>
</tbody>
</table>

is the specification for a tube having sides of 100mm x 60mm and the wall thickness of 3.6mm.
Tubes, like pipes can also be input by their dimensions (Height, Width and Thickness) instead of by their table
designations.

\[
\begin{array}{ccc}
6 & TA & ST \\
 & TUBE & DT \\
 & 8.0 & WT \\
 & 6.0 & TH \\
 & 0.5 & \\
\end{array}
\]

is a tube that has a height of 8, a width of 6, and a wall thickness of 0.5 in current length units. Only code
checking and no member selection will be performed for TUBE sections specified this way.

**D7.A.4.11 Example**

Sample input file containing German shapes:

```
STAAD SPACE
UNIT METER KN
JOINT COORDINATES
1 0 0 15 140 0 0
MEMBER INCIDENCES
1 1 2 14
UNIT CM
MEMBER PROPERTIES GERMAN
* IPE SHAPES
1 TA ST IPEA120
* HE SHAPES
2 TA ST HEB300
* I SHAPES
3 TA ST I200
* T SHAPES
4 TA T HEA220
* U CHANNELS
5 TA ST U70X40
* DOUBLE U CHANNELS
6 TA D U260
* ANGLES
7 TA ST L20X20X2.5
* REVERSE ANGLES
8 TA RA L40X20X5
* DOUBLE ANGLES - LONG LEGS BACK TO BACK
9 TA LD L40X20X4 SP 0.5
* DOUBLE ANGLES - SHORT LEGS BACK TO BACK
10 TA SD L40X20X4 SP 0.5
* PIPES
11 TA ST PIP602.9
* PIPES
12 TA ST PIPE OD 25.0 ID 20.0
* TUBES
13 TA ST TUB100603.6
* TUBES
14 TA ST TUBE DT 8.0 WT 6.0 WT 0.5
*
PRINT MEMBER PROPERTIES
FINISH
```

**D7.A.5 Member Capacities**

The allowable stresses used in the implementation are based on DIN 18800 (Part 1) - Section 7. The procedures
of DIN 18800 Part 2 are used for stability analysis. The basic measure of member capacities are the allowable
stresses on the member under various conditions of applied loading such as allowable tensile stress, allowable
compressive stress etc. These depend on several factors such as cross sectional properties, slenderness factors,
unsupported width to thickness ratios and so on. Explained here is the procedure adopted in STAAD for calculating such capacities.

**D7.A.5.1 Checks for Axial Tension**

In members with axial tension, the tensile load must not exceed the tension capacity of the member. The tension capacity of the member is calculated on the basis of the member area. STAAD calculates the tension capacity of a given member based on a user supplied net section factor (NSF - a default value of 1.0 is present but may be altered by changing the input value, see D7.A.7 Design Parameters (on page 1884)) and proceeds with member selection or code checking.

**D7.A.5.2 Checks for Axial Compression**

The compression capacity for members in compression is determined according to the procedure of DIN 18800-Part 2. Compressive resistance is a function of the slenderness of the cross-section (Kl/r ratio) and the user may control the slenderness value by modifying parameters such as KY, LY, KZ and LZ.

**D7.A.5.3 Checks for Bending and Shear**

The bending compressive and tensile capacities are dependent on such factors as length of outstanding legs, thickness of flanges, unsupported length of the compression flange (UNL, defaults to member length) etc. Shear capacities are a function of web depth, web thickness etc. Users may use a value of 1.0 or 2.0 for the TRACK parameter to obtain a listing of the bending and shear capacities.

**D7.A.6 Combined Loading**

For members experiencing combined loading (axial force, bending, and shear), applicable interaction formulas are checked at different locations of the member for all modeled loading situations. Members subjected to axial force and bending are checked using the criteria of DIN 18800 (Part 1) - Section 6.1.6. In addition, for members with axial loads and bending, the criteria of DIN 18800(Part 2) - Sections 3.4 and 3.5 are used.

**D7.A.7 Design Parameters**

You are allowed complete control over the design process through the use of parameters described in the following table. These parameters communicate design decisions from the engineer to the program. The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of the situation, some or all of these parameter values may have to be changed to exactly model the physical structure.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

**Table 164: German Steel Design Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td></td>
<td>Must be specified as DIN18800. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| BEAM           | 0.0           | Number of sections to be checked per member:  
|                |               | 0. Design only for end sections.  
|                |               | 1. Check at location of maximum MZ along member.  
|                |               | 2. Check ends plus location of beam 1.0 check.  
|                |               | 3. Check at every 1/13th of the member length and report the maximum. |
| CB             | 0             | Beam coefficient n, defined in Table 9: If Cb = 0, program will use n = 2.5 for rolled sections and 2.0 for welded sections. |
| CMM            | 1.0           | Moment factor, Zeta, defined in Table 10:  
|                |               | 1. fixed ended member with constant moment, Zeta = 1.0  
|                |               | 2. pin ended member with UDL, Zeta = 1.12  
|                |               | 3. pin ended member with central point load, Zeta = 1.35  
|                |               | 4. fixed ended member, Zeta calculated from end moments. |
| DMAX           | 1.0 m         | Maximum allowable depth during member selection |
| DMIN           | 0.0 m         | Minimum required depth during member selection |
| KY             | 1.0           | K value in local y-axis. Usually, this is the minor axis. |
| KZ             | 1.0           | K value in local z-axis. Usually, this is the major axis. |
| LY             | Member Length | Length in local y-axis to calculate slenderness ratio. |
| LZ             | Member Length | Length in local z-axis to calculate slenderness ratio. |
| PY             | 240 N/sq.mm   | Strength of steel. |
### D7.A.9 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate to carry the forces transmitted to it by the loads on the structure. The adequacy is checked per the DIN requirements.

Code checking is done using forces and moments at specified sections of the members. If the `BEAM` parameter for a member is set to 1, moments are calculated at every twelfth point along the beam, and the maximum moment about the major axis is used. When no sections are specified and the `BEAM` parameter is set to zero (default),

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual to allowable stresses</td>
</tr>
<tr>
<td>SAME</td>
<td>0.0</td>
<td>Control of sections to try during a SELECT process:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Try every section of the same type as the original.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Try only those with a similar name.</td>
</tr>
<tr>
<td>SBLT</td>
<td>0</td>
<td>Specify section as either rolled or built-up:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Rolled</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Built-up</td>
</tr>
<tr>
<td>SGR</td>
<td>0.0</td>
<td>Grade of steel:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. St 37-2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. St 52-3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. St E 355</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Level of detail in output file:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Output summary of results</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Output summary of results plus member capacities</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Output detailed results</td>
</tr>
<tr>
<td>UNF</td>
<td>1.0</td>
<td>Same as above provided as a factor of actual member length.</td>
</tr>
<tr>
<td>UNL</td>
<td>Member Length</td>
<td>Unrestrained member length in lateral torsional buckling checks.</td>
</tr>
</tbody>
</table>
design will be based on member start and end forces. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from start joint) and magnitudes of the governing forces and moments are also printed.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

D7.A.9 Member Selection

The member selection process basically involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on TUBES, PIPES, or members listed as PRISMATIC.

```
UNIT METER
PARAMETER
CODE GERMAN
NSF 0.85 ALL
UNL 10.0 MEMBER 7
KY 1.2 MEMBER 3 4
RATIO 0.9 ALL
TRACK 1.0 ALL
CHECK CODE ALL
```

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

D8. Indian Codes

D8.A. Indian Codes - Concrete Design per IS 456

STAAD.Pro is capable of performing concrete design based on the Indian code IS 456 2000 Code of Practice for Plain and Reinforced Concrete.

Related Links
- D8.A. Indian Codes - Concrete Design per IS 456 (on page 1887)

D8.A.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams — Prismatic (Rectangular & Square), T-Beams, and L-shapes
- For Columns — Prismatic (Rectangular, Square, and Circular)
D8.A.2 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the MEMBER PROPERTY command. The following example shows the required input:

```
UNIT MM
MEMBER PROPERTY
  1 3 TO 7 9 PRISM YD 450. ZD 250.
  11 13 PR YD 350.
  14 TO 16 PRIS YD 400. ZD 750. YB 300. ZB 200.
```

will be done accordingly. In the above input, the first set of members are rectangular (450 mm depth and 250 mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 350 mm diameter. The third set numbers in the above example represents a T-shape with 750 mm flange width, 200 width, 400 mm overall depth and 100 mm flange depth (See section 6.20.2). The program will determine whether the section is rectangular, flanged or circular and the beam or column design.

D8.A.3 Design Parameters

The program contains a number of parameters which are needed to perform design as per IS:456 2000. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 9A.1 of this manual contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

**Table 165: Indian Concrete Design IS456 2000 Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as INDIAN. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2859).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| BRACING        | 0.0          | Beam Design:  
|                |              | A value of 1.0 means the effect of axial force will be taken into account for beam design.  
|                |              | Column Design:  
|                |              | correspond to the terms “Braced” and “Unbraced” described in Notes 1, 2, and 3 of Clause 39.7.1 of IS456:2000.  
|                |              | 0) The column is braced about both axes.  
|                |              | 1) The column is unbraced about major axis.  
|                |              | 2) The column is unbraced about minor axis.  
|                |              | 3) The column is unbraced about both axis.  |
| CLB            | 25 mm        | The clear cover for the outermost bottom reinforcement in beams.  
|                |              | Tip: If you want to specify the clear distance to ties or stirrups, include the full diameter of those in this value. For example, beams with 25 mm clear and 8 mm ties should use CLB 33. |
| CLEAR          | 30 mm (beams) 40 mm (columns) | The clear distance between to the main member reinforcement.  
<p>|                |              | Tip: If you want to specify the clear distance to ties or stirrups, include the full diameter of those in this value. For example, columns with 35 mm clear and 8 mm ties should use CLEAR 43. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CLT</td>
<td>25 mm</td>
<td>The clear cover for the outermost top reinforcement in beams. <strong>Tip:</strong> If you want to specify the clear distance to ties or stirrups, include the full diameter of those in this value. For example, beams with 25 mm clear and 8 mm ties should use CLT 33.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>Total depth to be used for design. This value defaults to YD as provided under MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>EFACE</td>
<td>0.0</td>
<td>Face of support location at end of beam. The parameter can also be used to check against shear at any point from the end of the member. <strong>Note:</strong> Both SFACE and EFACE are input as positive numbers.</td>
</tr>
<tr>
<td>ELZ</td>
<td>1.0</td>
<td>Ratio of effective length to actual length of column about major axis. See Note b below.</td>
</tr>
<tr>
<td>ELY</td>
<td>1.0</td>
<td>Ratio of effective length to actual length of column about minor axis. See Note b below.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| ENSH           | 0.0           | Perform shear check against enhanced shear strength as per Cl. 40.5 of IS456:2000.  
                   • ENSH = 1.0 means ordinary shear check to be performed (no enhancement of shear strength at sections close to support)  
                   • For ENSH = a positive value (say x), shear strength will be enhanced up to a distance x from the start of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note.)  
                   • For ENSH = a negative value (say -y), shear strength will be enhanced up to a distance y from the end of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note.)  
                   For default value (0.0) is used the program will calculate Length to Overall Depth ratio. If this ratio is greater than 2.5, shear strength will be enhanced at sections (<2d) close to support otherwise ordinary shear check will be performed. |
<p>| FC             | 30 N/mm²      | Concrete Yield Stress. |
| FYMAIN         | 415 N/mm²     | Yield Stress for main reinforcing steel. |
| FYSEC          | 415 N/mm²     | Yield Stress for secondary reinforcing steel. |
| MINMAIN        | 10 mm         | Minimum main reinforcement bar size. |
| MAXMAIN        | 60 mm         | Maximum main reinforcement bar size. |
| MINSEC         | 8 mm          | Minimum secondary reinforcement bar size. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAXSEC</td>
<td>12 mm</td>
<td>Maximum secondary reinforcement bar size.</td>
</tr>
</tbody>
</table>
| MFACE          | 0             | Design beam for flexure at any point along the length of the beam as specified by SFACE and EFACE parameters.  
0) (Off) Do not design at sections  
1) (On) Design at specified sections from start and end of the members assigned.  
**Note**: If SFACE and EFACE sections are *not* previously defined, then the output will present a warning and no flexure design will be performed (i.e., the MFACE parameter is ignored). |
| RATIO          | 4.0           | Maximum percentage of longitudinal reinforcement in columns. |
| REINF          | 0.0           | Tied column. A value of 1.0 will mean spiral reinforcement. |
| RENSH          | 0.0           | Distance of the start or end point of the member from its nearest support. This parameter is used only when a span of a beam is subdivided into two or more parts.  
(Refer note) |
| RFACE          | 4             | 2) Two faced distribution about major axis.  
3) Two faced distribution about minor axis.  
4) Longitudinal reinforcement in column is arranged equally along 4 faces. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SFACE</td>
<td>0.0</td>
<td>Face of support location at start of beam. It is used to check against shear at the face of the support in beam design. The parameter can also be used to check against shear at any point from the start of the member.</td>
</tr>
<tr>
<td>SPSMAIN</td>
<td>25 mm</td>
<td>Minimum clear distance between main reinforcing bars in beam and column. For column center to center distance between main bars cannot exceed 300 mm.</td>
</tr>
<tr>
<td>TORSION</td>
<td>0.0</td>
<td>0) torsion to be considered in beam design. 1) torsion to be neglected in beam design.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>Beam Design: 0) output consists of reinforcement details at START, MIDDLE, and END. 1) critical moments are printed in addition to TRACK 0.0 output. 2) required steel for intermediate sections defined by NSECTION are printed in addition to TRACK 1.0 output. Column Design: 0) reinforcement details are printed. 1) column interaction analysis results are printed in addition to TRACK 0.0 output. 2) a schematic interaction diagram and intermediate interaction values are printed in addition to TRACK 1.0 output.</td>
</tr>
<tr>
<td>ULY</td>
<td>1.0</td>
<td>Ratio of unsupported length to actual length of column about minor axis. See Note c below.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>ULZ</td>
<td>1.0</td>
<td>Ratio of unsupported length to actual length of column about major axis. See Note c below.</td>
</tr>
<tr>
<td>WIDTH</td>
<td>ZD</td>
<td>Width to be used for design. This value defaults to ZD as provided under MEMBER PROPERTIES.</td>
</tr>
</tbody>
</table>

**Notes**

a. You may specify reinforcing bar combinations through the BAR COMBINATION command. Refer to D8.A.7 Bar Combination (on page 1899) for details.

b. ELY and ELZ parameters are used to calculate effective length of column to find whether it is a short or long column. Please refer CL 25.1.2 of IS456:2000.

In CL 25.1.2 of IS456:2000, you will find two term, \( l_{ex} \) and \( l_{ey} \), which STAAD.Pro calculates as:

- \( l_{ex} = \) ELZ multiplied by the member length (distance between the two nodes of the member)
- \( l_{ey} = \) ELY multiplied by the member length (distance between the two nodes of the member)

For the term "D" in CL 25.1.2 of IS456:2000, STAAD.Pro uses the YD dimension of the column.

For the term "b" in CL 25.1.2 of IS456:2000, STAAD.Pro uses the ZD dimension of the column.

c. ULY and ULZ parameters are used to calculate unsupported length of column to find minimum eccentricity. Please refer CL 25.4 of IS456:2000.

In CL 25.4 of IS456:2000, you will find an expression "unsupported length of column". This term is calculated as

- \( ULZ \) multiplied by the member length for the Z axis
- \( ULY \) multiplied by the member length for the Y axis

d. The value of the ENSH parameter (other than 0.0 and 1.0) is used only when the span of a beam is subdivided into two or more parts. When this condition occurs, the RENV parameter is also to be used.

The span of the beam is subdivided four parts, each of length L meter. The shear strength will be enhanced up to X meter from both supports. The input should be the following:

**Steps:**

a. **ENSH L MEMB 1 =>** Shear strength will be enhanced throughout the length of the member 1, positive sign indicates length measured from start of the member
b. $\text{ENSH (X-L) MEMB 2} =>$ Shear strength will be enhanced up to a length (X-L) of the member 2, length measured from the start of the member

c. $\text{ENSH \text{-L MEMB 4} =>}$ Shear strength will be enhanced throughout the length of the member 4, negative sign indicates length measured from end of the member

d. $\text{ENSH \text{-(X-L) MEMB 3} =>}$ Shear strength will be enhanced up to a length (X-L) of the member 3, length measured from the end of the member

e. $\text{RENSH L MEMB 2 3} =>$ Nearest support lies at a distance L from both the members 2 and 3.

f. $\text{DESIGN BEAM 1 TO 4} =>$ This will enhance the shear strength up to length X from both ends of the beam consisting of members 1 to 4 and gives spacing accordingly.

At section = $y_1$ from start of member 1, $av = y_1$
At section = $y_2$ from the start of member 2, $av = y_2 + L$
At section = $y_3$ from the end of member 3, $av = y_3 + L$
At section = $y_4$ from end of member 4, $av = y_4$

where $tc, \text{enhanced} = \frac{2dtc}{av}$

At section 0.0, av becomes zero. Thus enhanced shear strength will become infinity. However for any section shear stress cannot exceed $tc, \text{max}$. Hence enhanced shear strength is limited to a maximum value of $tc, \text{max}$.

D8.A.4 Slenderness Effects and Analysis Consideration

Slenderness effects are extremely important in designing compression members. The IS:456 code specifies two options by which the slenderness effect can be accommodated (Clause 39.7). One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed end moments, the effect of deflections on moment and forces and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD has been written to allow the use of the first option. To perform this type of analysis, use the command $\text{PDELTA ANALYSIS}$ instead of $\text{PERFORM ANALYSIS}$. The P-Delta analysis will accommodate all requirements of the second-order analysis described by IS:456, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, additional moments are calculated based on empirical formula and assumptions on sidesway (Clause 39.7.1 and 39.7.1.1,IS:456 - 2000). The rules of Clause 39.7.1 have been implemented in STAAD.Pro. They will be checked if the ELV and ELZ parameters are specified.

Considering all these information, a P-Delta analysis, as performed by STAAD may be used for the design of concrete members.

**Note:** To take advantage of this analysis, all the combinations of loading must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments (i.e., analysis results), whereas a primary load case is revised during the P-delta analysis based on the deflections. Loads can be combined prior to analysis using the $\text{REPEAT LOAD}$ command.

**Note:** You must specify the appropriate load factors (e.g., 1.5 for dead load, etc.) as STAAD does not factor the loads automatically.
D8.A.5 Beam Design

Beams are designed for flexure, shear and torsion. If required the effect the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The design is performed at 13 evenly spaced points along the length of the beam, including start and end points (i.e., 1/12th points or at ends of 12 equal length segments). All of these sections are scanned to determine the design force envelopes.

![Diagram of beam sections](image)

**Figure 183: The sections used for design along the length of a member**

**D8.A.5.1 Design for Flexure**

Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the above mentioned sections. Each of these sections is designed to resist both of these critical sagging and hogging moments. Where ever the rectangular section is inadequate as singly reinforced section, doubly reinforced section is tried. However, presently the flanged section is designed only as singly reinforced section under sagging moment. It may also be noted all flanged sections are automatically designed as rectangular section under hogging moment as the flange of the beam is ineffective under hogging moment. Flexural design of beams is performed in two passes. In the first pass, effective depths of the sections are determined with the assumption of single layer of assumed reinforcement and reinforcement requirements are calculated. After the preliminary design, reinforcing bars are chosen from the internal database in single or multiple layers. The entire flexure design is performed again in a second pass taking into account of the changed effective depths of sections calculated on the basis of reinforcement provide after the preliminary design. Final provisions of flexural reinforcements are made then. Efforts have been made to meet the guideline for the curtailment of reinforcements as per IS:456-2000 (Clause 26.2.3). Although exact curtailment lengths are not mentioned explicitly in the design output (finally which will be more or less guided by the detailer taking into account of other practical consideration), user has the choice of printing reinforcements provided by STAAD at 11 equally spaced sections from which the final detail drawing can be prepared.

Once you have specified SFACE and EFACE parameters to indicate sections, the MFACE parameter can be used to design for flexure at any point along the length of the beam, in addition to the equally spaced sections normally used.

**D8.A.5.2 Design for Shear**

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear design are performed at 11 equally spaced sections (0.0 to 1.0) for the maximum shear forces amongst the active load cases and the associated torsional moments. Shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by STAAD program. Two-legged stirrups are provided to take care of the balance shear forces acting on these sections.

As per Clause 40.5 of IS:456-2000 shear strength of sections (< 2d where d is the effective depth) close to support has been enhanced, subjected to a maximum value of \( \tau_{\text{max}} \).
**D8.A.5.3 Beam Design Output**

The default design output of the beam contains flexural and shear reinforcement provided at 5 equally spaced (0, .25, .5, .75 and 1.) sections along the length of the beam. User has option to get a more detail output. All beam design outputs are given in IS units. An example of rectangular beam design output with TRACK 2.0 output is presented below:

<table>
<thead>
<tr>
<th>BEAM NO.</th>
<th>DESIGN RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>M20</td>
<td>Fe415 (Main)</td>
</tr>
<tr>
<td></td>
<td>Fe250 (Sec.)</td>
</tr>
<tr>
<td>LENGTH:</td>
<td>6400.0 mm</td>
</tr>
<tr>
<td>SIZE:</td>
<td>300.0 mm x 400.0 mm</td>
</tr>
<tr>
<td>COVER:</td>
<td>25.0 mm</td>
</tr>
</tbody>
</table>

**DESIGN LOAD SUMMARY (KN MET)**

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>FLEXURE (Maxm. Sagging/Hogging moments)</th>
<th>SHEAR</th>
<th>MX Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.00/0.00/0.00/0.00/0.00 1</td>
<td>60.61</td>
<td>0.00/1</td>
</tr>
<tr>
<td>533.3</td>
<td>0.00/29.63/0.00/0.00/0.00 1</td>
<td>50.51</td>
<td>0.00/1</td>
</tr>
<tr>
<td>1066.7</td>
<td>0.00/53.88/0.00/0.00/0.00 1</td>
<td>40.41</td>
<td>0.00/1</td>
</tr>
<tr>
<td>1600.0</td>
<td>0.00/72.73/0.00/0.00/0.00 1</td>
<td>30.31</td>
<td>0.00/1</td>
</tr>
<tr>
<td>2133.3</td>
<td>0.00/86.20/0.00/0.00/0.00 1</td>
<td>20.20</td>
<td>0.00/1</td>
</tr>
<tr>
<td>2666.7</td>
<td>0.00/94.28/0.00/0.00/0.00 1</td>
<td>10.10</td>
<td>0.00/1</td>
</tr>
<tr>
<td>3200.0</td>
<td>0.00/96.98/0.00/0.00/0.00 1</td>
<td>0.00</td>
<td>0.00/1</td>
</tr>
<tr>
<td>3733.3</td>
<td>0.00/94.28/0.00/0.00/0.00 1</td>
<td>-10.10</td>
<td>0.00/1</td>
</tr>
<tr>
<td>4266.7</td>
<td>0.00/86.20/0.00/0.00/0.00 1</td>
<td>-20.20</td>
<td>0.00/1</td>
</tr>
<tr>
<td>4800.0</td>
<td>0.00/72.73/0.00/0.00/0.00 1</td>
<td>-30.31</td>
<td>0.00/1</td>
</tr>
<tr>
<td>5333.3</td>
<td>0.00/53.88/0.00/0.00/0.00 1</td>
<td>-40.41</td>
<td>0.00/1</td>
</tr>
<tr>
<td>5866.7</td>
<td>0.00/29.63/0.00/0.00/0.00 1</td>
<td>-50.51</td>
<td>0.00/1</td>
</tr>
<tr>
<td>6400.0</td>
<td>0.00/0.00/0.00/0.00/0.00 1</td>
<td>-60.61</td>
<td>0.00/1</td>
</tr>
</tbody>
</table>

**SUMMARY OF REINF. AREA (Sq.mm)**

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>TOP Reqd./Provided reinf.</th>
<th>BOTTOM Reqd./Provided reinf.</th>
<th>STIRRUPS (2 legged)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.00/402.12 (2-16)</td>
<td>0.00/981.75 (2-25)</td>
<td>8@180 mm</td>
</tr>
<tr>
<td>533.3</td>
<td>0.00/402.12 (2-16)</td>
<td>237.32/1472.62 (3-25)</td>
<td>8@180 mm</td>
</tr>
<tr>
<td>1066.7</td>
<td>0.00/402.12 (2-16)</td>
<td>450.84/1472.62 (3-25)</td>
<td>8@180 mm</td>
</tr>
<tr>
<td>1600.0</td>
<td>0.00/402.12 (2-16)</td>
<td>632.82/1472.62 (3-25)</td>
<td>8@180 mm</td>
</tr>
</tbody>
</table>
D8.A.6 Column Design

Columns are designed for axial forces and biaxial moments at the ends. All active load cases are tested to calculate reinforcement. The loading which yield maximum reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. By default, square and rectangular columns and designed with reinforcement distributed on each side equally for the sections under biaxial moments and with reinforcement distributed equally in two faces for sections under uniaxial moment. User may change the default arrangement of the reinforcement with the help of the parameter RFACE (see Table 8A.1). Depending upon the member lengths, section dimensions and effective length coefficients specified by the user STAAD automatically determine the criterion (short or long) of the column design. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS:456 have been taken care of in the column design of STAAD. Default clear spacing between main reinforcing bars is taken to be 25 mm while arrangement of longitudinal bars.

Default column design output (TRACK 0.0) contains the reinforcement provided by STAAD and the capacity of the section. With the option TRACK 1.0, the output contains intermediate results such as the design forces, effective length coefficients, additional moments etc. All design output is given in SI units. An example of a TRACK 2.0 output follows:

<table>
<thead>
<tr>
<th>COLUMN NO.</th>
<th>DESIGN RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>M20</td>
<td></td>
</tr>
<tr>
<td>LENGTH: 3000.0 mm</td>
<td>CROSS SECTION: 400.0 mm X 600.0 mm</td>
</tr>
<tr>
<td>** GUIDING LOAD CASE: 1 END JOINT: 1 SHORT COLUMN**</td>
<td></td>
</tr>
<tr>
<td>DESIGN FORCES (KNS-MET)</td>
<td></td>
</tr>
<tr>
<td>DESIGN AXIAL FORCE (Pu) :</td>
<td>2000.00</td>
</tr>
<tr>
<td>ABOUT Z</td>
<td>160.00</td>
</tr>
<tr>
<td>ABOUT Y</td>
<td>52.00</td>
</tr>
<tr>
<td>SLENDERNESS RATIOS</td>
<td>-</td>
</tr>
<tr>
<td>MOMENTS DUE TO SLENDERNESS EFFECT</td>
<td>-</td>
</tr>
<tr>
<td>MOMENT REDUCTION FACTORS</td>
<td>-</td>
</tr>
<tr>
<td>ADDITION MOMENTS (Maz and May)</td>
<td>-</td>
</tr>
<tr>
<td>TOTAL DESIGN MOMENTS</td>
<td>160.00</td>
</tr>
<tr>
<td>REQD. STEEL AREA :</td>
<td>3587.44 Sq.mm.</td>
</tr>
<tr>
<td>REQD. CONCRETE AREA:</td>
<td>236412.56 Sq.mm.</td>
</tr>
<tr>
<td>MAIN REINFORCEMENT : Provide 32 - 12 dia. (1.51%, 3619.11 Sq.mm.) (Equally distributed)</td>
<td></td>
</tr>
<tr>
<td>TIE REINFORCEMENT : Provide 8 mm dia. rectangular ties @ 190 mm c/c</td>
<td></td>
</tr>
</tbody>
</table>
SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
----------------------------------------------------------
Puz : 3244.31   Muz1 : 269.59   Muy1 : 168.42

INTERACTION RATIO: 0.98 (as per Cl. 39.6, IS456:2000)

SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
----------------------------------------------------------
WORST LOAD CASE: 1
END JOINT: 1 Puz : 3253.88   Muz : 271.48   Muy : 170.09   IR: 0.96
============================================================================

D8.A.7 Bar Combination

Initially the program selects only one bar to calculate the number of bars required and area of steel provided at each section along the length of the beam. You may use the BAR COMBINATION command to specify two bar diameters to calculate a combination of each bar to be provided at each section. The syntax for bar combination is given below.

START  BAR COMBINATION
MD1 <bar diameter> MEMB <member list>
MD2 <bar diameter> MEMB <member list>
END  BAR COMBINATION

Note: The bar sizes should be specified in the order of increasing size (i.e., MD2 bar diameter should be greater than MD1 bar diameter).

The beam length is divided into three parts, two at its ends and one at span. Ld gives the development length to be provided at the two ends of each section.

The typical output for bar combination is shown below:

<table>
<thead>
<tr>
<th>M A I N         R E I N F O R C E M E N T</th>
</tr>
</thead>
<tbody>
<tr>
<td>SECTION</td>
</tr>
<tr>
<td>TOP</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Ast Reqd</td>
</tr>
<tr>
<td>Prov</td>
</tr>
</tbody>
</table>

STAAD.Pro 1899 User Manual
D8.A.8 Wall Design in accordance with IS 456-2000

**Note:** Shearwall design has been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow it but its use is not recommended.

D8.A.9 Element Design

Element design will be performed only for the moments MX and MY at the center of the element. Design will not be performed for SX, SY, SXY, SQX, SQY or MXY. Also, design is not performed at any other point on the surface of the element.

A typical example of element design output is shown below. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement (Refer to D8.A.1 Section Types for Concrete Design (on page 1887)). The parameters FYMAIN, FC, and CLEAR listed in D8.A.3 Design Parameters (on page 1888) are relevant to slab design. Other parameters mentioned in Table 11A.1 are not applicable to slab design.

![Figure 184: Sign convention of loaded plate element](image)

**Example Element Design Output**

```plaintext
ELEMENT DESIGN SUMMARY

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LONG. REINF</th>
<th>MOM-X /LOAD</th>
<th>TRANS. REINF</th>
<th>MOM-Y /LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
```

STAAD.Pro 1900 User Manual
D8.B. Indian Codes - Concrete Design per IS 13920-1993

STAAD.Pro is capable of performing concrete design based on the 1993 edition of the Indian code IS 13920 *Code of Practice for Ductile Detailing of Reinforced Concrete Structures Subjected to Seismic Forces*. Designs per IS 13920-1993 satisfy all provisions of IS 456 - 2000 and IS 13920-1993 for beams and columns (Refer to D8.A, Indian Codes - Concrete Design per IS 456 (on page 1887)).

D8.B.1 Design Operations

Earthquake motion often induces force large enough to cause inelastic deformations in the structure. If the structure is brittle, sudden failure could occur. But if the structure is made to behave ductile, it will be able to sustain the earthquake effects better with some deflection larger than the yield deflection by absorption of energy. Therefore ductility is also required as an essential element for safety from sudden collapse during severe shocks.

D8.B.2 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams: Prismatic (Rectangular & Square) and T-shape
- For Columns : Prismatic (Rectangular, Square, and Circular)

D8.B.3 Design Parameters

The program contains a number of parameters that are needed to perform design as per IS 13920 1993. It accepts all parameters that are needed to perform design as per IS:456. Over and above it has some other parameters that are required only when designed is performed as per IS:13920 1993. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. The following table contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.
## Table 166: Indian Concrete Design IS 13920 1993 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td></td>
<td>Must be specified as IS13920 1993 Design code to follow. See <a href="#">TR.53.2 Concrete Design-Parameter Specification</a> (on page 2859).</td>
</tr>
<tr>
<td>BRACING</td>
<td>0.0</td>
<td>Beam Design</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = the effect of axial force will be taken into account for beam design.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Column Design: Correspond to the terms &quot;Braced&quot; and &quot;Unbraced&quot; described in Notes 1, 2, and 3 of Clause 39.7.1 of IS456:2000.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = the column is unbraced about major axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.0 = the column is unbraced about minor axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3.0 = the column is unbraced about both axis.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>Total depth to be used for design. This value defaults to YD (depth of section in Y direction) as provided under MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>CLEAR</td>
<td>25 mm, 40 mm</td>
<td>For beam members. For column members</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> This is the clear cover to the outermost main reinforcing bar. It is <em>not</em> the clear cover for the stirrups or the tie bars.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| COMBINE        | 0.0          | Default value means there will be no member combination.  
1.0 = no printout of sectional force and critical load for combined member in the output.  
2.0 = printout of sectional force for combined member in the output.  
3.0 = printout of both sectional force and critical load for combined member in the output.*** |
| EFACE          | 0.0          | Face of support location at end of beam. The parameter can also be used to check against shear at any point from the end of the member.  
**Note:** Both SFACE and EFACE are input as positive numbers.* |
<p>| ELZ            | 1.0          | Ratio of effective length to actual length of column about major axis. |
| ELY            | 1.0          | Ratio of effective length to actual length of column about minor axis. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| ENSH           | 0.0           | Perform shear check against enhanced shear strength as per Cl. 40.5 of IS456:2000.  
1.0 = ordinary shear check to be performed (no enhancement of shear strength at sections close to support)  
a positive value (say x) = shear strength will be enhanced up to a distance x from the start of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note after Table 8A.1)  
a negative value (say -y) = shear strength will be enhanced up to a distance y from the end of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note after Table 8A.1)  
0.0 = the program will calculate Length to Overall Depth ratio. If this ratio is greater than 2.5, shear strength will be enhanced at sections (<2d) close to support otherwise ordinary shear check will be performed. |
<p>| EUDL           | None          | Equivalent u.d.l on span of the beam. This load value must be the unfactored load on span. During design the load value is multiplied by a factor 1.2. If no u.d.l is defined factored shear force due to gravity load on span will be taken as zero. No elastic or plastic moment will be calculated. Shear design will be performed based on analysis result. (Refer note) |
| FYMAIN         | 415 N/mm²     | Yield Stress for main reinforcing steel. |
| FYSEC          | 415 N/mm²     | Yield Stress for secondary reinforcing steel. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FC</td>
<td>30 N/mm$^2$</td>
<td>Concrete Yield Stress.</td>
</tr>
<tr>
<td>GLD</td>
<td>None</td>
<td>Gravity load number to be considered for calculating equivalent u.d.l on span of the beam, in case no EUDL is mentioned in the input. This load case can be any static load case containing MEMBER LOAD on the beam which includes UNI, CON, LIN and TRAP member loading. CMOM member loading is considered only when it is specified in local direction. FLOOR LOAD is also considered. The load can be primary or combination load. For combination load only load numbers included in load combination is considered. The load factors are ignored. Internally the unfactored load is multiplied by a factor 1.2 during design. If both EUDL and GLD parameters are mentioned in the input mentioned EUDL will be considered in design.</td>
</tr>
<tr>
<td>HLINK</td>
<td>Spacing of longitudinal bars measured to the outer face</td>
<td>Longer dimension of the rectangular confining hoop measured to its outer face. It shall not exceed 300 mm as per Cl. 7.4.8. If the HLINK value as provided in the input file does not satisfy the clause the value will be internally assumed as the default one. This parameter is valid for rectangular column.</td>
</tr>
</tbody>
</table>

Note: No dynamic (Response spectrum, 1893, Time History) and moving load cases are considered. CMOM member loading in global direction is not considered. UMOM member loading is not considered.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| IPLM           | 0.0           | Default value calculates elastic/plastic hogging and sagging moments of resistance of beam at its ends.  
1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at start node of beam. This implies no support exists at start node.  
-1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at start node of beam. This implies support exists at start node.  
2.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at end node of beam. This implies no support exists at end node.  
-2.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at end node of beam. This implies support exists at end node. ** |
| IMB            | 0.0           | Default value calculates elastic/plastic hogging and sagging moments of resistance of beam at its ends.  
1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at both ends of beam. This implies no support exist at either end of the member.  
-1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at both ends of beam. This implies support exist at both ends of the member. ** |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MINMAIN</td>
<td>10 mm</td>
<td>Minimum main reinforcement bar size.</td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>60 mm</td>
<td>Maximum main reinforcement bar size.</td>
</tr>
<tr>
<td>MINSEC</td>
<td>8 mm</td>
<td>Minimum secondary reinforcement bar size.</td>
</tr>
<tr>
<td>MAXSEC</td>
<td>12 mm</td>
<td>Maximum secondary reinforcement bar size.</td>
</tr>
</tbody>
</table>
| PLASTIC        | 0.0           | Default value calculates elastic hogging and sagging moments of resistance of beam at its ends.  
1.0 = plastic hogging and sagging moments of resistance of beam to be calculated at its ends. |
| RATIO          | 4.0           | Maximum percentage of longitudinal reinforcement in columns. |
| REINF          | 0.0           | 0.0 = Tied column (default)  
1.0 = spiral reinforcement |
| RENSH          | 0.0           | Distance of the start or end point of the member from its nearest support. This parameter is used only when a span of a beam is subdivided into two or more parts.  
Refer to Notes (on page 1894) for IS456 Design Parameters. |
| RFACE          | 4.0           | 4.0 = longitudinal reinforcement in column is arranged equally along four faces.  
2.0 invokes two faced distribution about major axis.  
3.0 invokes two faced distribution about minor axis. |
### Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>SFACE</strong></td>
<td>0.0</td>
<td>Face of support location at start of beam. It is used to check against shear at the face of the support in beam design. The parameter can also be used to check against shear at any point from the start of the member.*</td>
</tr>
<tr>
<td><strong>SPSMAIN</strong></td>
<td>25 mm</td>
<td>Minimum clear distance between main reinforcing bars in beam and column. For column center to center distance between main bars cannot exceed 300 mm.</td>
</tr>
<tr>
<td><strong>TORISION</strong></td>
<td>0.0</td>
<td>0.0 = torsion to be considered in beam design. 1.0 = torsion to be neglected in beam design.</td>
</tr>
</tbody>
</table>
| **TRACK**      | 0.0           | Beam Design: 0) output consists of reinforcement details at START, MIDDLE and END. 1) critical moments are printed in addition to TRACK  0 output. 2) required steel for intermediate sections defined by NSECTION are printed in addition to TRACK  1 output.  
Column Design: 0) reinforcement details are printed. 1) column interaction analysis results are printed in addition to TRACK  0 output. 2) a schematic interaction diagram and intermediate interaction values are printed in addition to TRACK  1 output. |
Bar combination has been introduced for detailing. Please refer D8.B.6 Bar Combination (on page 1915) for details.

* EFACE and SFACE command is not valid for member combination.

** IPLM and IMB commands are not valid for member combination. These commands are ignored for members forming physical member.

*** The purpose of COMBINE command is the following:

1. If a beam spanning between two supports is subdivided into many sub-beams this parameter will combine them into one member. It can also be used to combine members to form one continuous beam spanning over more than two supports.
2. When two or more members are combined during design plastic or elastic moments will be calculated at the column supports. At all the intermediate nodes (if any) this calculation will be ignored.

Note: Please note that the program only recognizes column at right angle to the beam. Inclined column support is ignored.

3. It will calculate sectional forces at 13 sections along the length of the combined member.
4. It will calculate critical loads (similar to that of Design Load Summary) for all active load cases during design. Beams will be combined only when DESIGN BEAM command is issued.

The following lines should be satisfied during combination of members:

1. Members to be combined should have same sectional properties if any single span between two column supports of a continuous beam is subdivided into several members.
2. Members to be combined should have same constants (E, Poi ratio, alpha, density, and beta angle)
3. Members to be combined should lie in one straight line.
4. Members to be combined should be continuous.
5. Vertical members (i.e., columns) cannot be combined.
6. Same member cannot be used more than once to form two different combined members.
7. The maximum number of members that can be combined into one member is 299.

Note: Sectional forces and critical load for combined member output will only be available when all the members combined are successfully designed in both flexure and shear.

ENSH and RENSH parameters will have to be provided (as and when necessary) even if physical member has been formed.
**D8.B.3.1 Example**

The following lines show a standard example for design to be performed in IS 13920 1993.

```
STAAD SPACE
UNIT METER MTON
JOINT COORDINATES

... 
MEMBER INCIDENCES 
... 
MEMBER PROPERTY INDIAN 
... 
CONSTANTS 
... 
SUPPORTS 
... 
DEFINE 1893 LOAD
ZONE 0.05 I 1 K 1 B 1
SELFWEIGHT
JOINT WEIGHT

... 
LOAD 1 SEISMIC LOAD IN X DIR
1893 LOAD X 1
LOAD 2 SEISMIC LOAD IN Z DIR
1893 LOAD Z 1
LOAD 3 DL
MEMBER LOAD
... UNI GY -5
LOAD 4 LL
MEMBER LOAD
... UNI GY -3
LOAD COMB 5 1.5(DL+LL)
3 1.5 4 1.5
LOAD COMB 6 1.2(DL+LL+SLX)
1 1.2 3 1.2 4 1.2
LOAD COMB 7 1.2(DL+LL-SLX)
1 1.2 3 1.2 4 -1.2
LOAD COMB 8 1.2(DL+LL+SLZ)
2 1.2 3 1.2 4 1.2
LOAD COMB 9 1.2(DL+LL-SLZ)
2 1.2 3 1.2 4 -1.2
PDELTA ANALYSIS
LOAD LIST 5 TO 9
START CONCRETE DESIGN
CODE IS13920 1993
UNIT MMS NEWTON
FYMAIN 415 ALL
FC 20 ALL
MINMAIN 12 ALL
MAXMAIN 25 ALL
TRACK 2.0 ALL
*** Unfactored gravity load on members 110 to 112 is 8 t/m (DL+LL) i.e.,
78.46 New/mm
EUDL 78.46 MEMB 110 TO 112
** Members to be combined into one physical member
COMBINE 3.0 MEMB 110 TO 112
*** Plastic moment considered
PLASTIC 1.0 MEMB 110 TO 112
```
**D8.B.4 Beam Design**

Beams are designed for flexure, shear and torsion. If required the effect of the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The total number of sections considered is 13. All of these sections are scanned to determine the design force envelopes.

For design to be performed as per IS:13920 the width of the member shall not be less than 200 mm (Clause 6.1.3). Also the member shall preferably have a width-to depth ratio of more than 0.3 (Clause 6.1.2).

The factored axial stress on the member should not exceed 0.1 fck (Clause 6.1.1) for all active load cases. If it exceeds allowable axial stress no design will be performed.

**D8.B.4.1 Design for Flexure**

Design procedure is same as that for IS 456. However while designing following criteria are satisfied as per IS-13920:

1. The minimum grade of concrete shall preferably be M20. (Clause 5.2)
2. Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3)
3. The minimum tension steel ratio on any face, at any section, is given by (Clause 6.2.1b)
   \[
   \rho_{\text{min}} = \frac{0.24 f_{ck}}{f_y}
   \]
   The maximum steel ratio on any face, at any section, is given by (Clause 6.2.2)
   \[
   \rho_{\text{max}} = 0.025
   \]
4. The positive steel ratio at a joint face must be at least equal to half the negative steel at that face. (Clause 6.2.3)
5. The steel provided at each of the top and bottom face, at any section, shall at least be equal to one-fourth of the maximum negative moment steel provided at the face of either joint. (Clause 6.2.4)

**D8.B.4.2 Design for Shear**

The shear force to be resisted by vertical hoops is guided by the Clause 6.3.3 of IS 13920:1993 revision. Elastic sagging and hogging moments of resistance of the beam section at ends are considered while calculating shear force. Plastic sagging and hogging moments of resistance can also be considered for shear design if PLASTIC parameter is mentioned in the input file. (Refer Table 8A1.1)

Shear reinforcement is calculated to resist both shear forces and torsional moments. Procedure is same as that of IS 456.

The following criteria are satisfied while performing design for shear as per Cl. 6.3.5 of IS-13920:

- The spacing of vertical hoops over a length of 2d at either end of the beam shall not exceed:
  - a. \( d/4 \)
  - b. 8 times the diameter of the longitudinal bars

In no case this spacing is less than 100 mm.
The spacing calculated from above, if less than that calculated from IS 456 consideration is provided.

**D8.B.4.3 Beam Design Output**

The default design output of the beam contains flexural and shear reinforcement provided at 5 equally spaced sections along the length of the beam. User has option to get a more detail output. All beam design outputs are given in IS units. An example of rectangular beam design output with the TRACK 2.0 is presented below:

```
B E A M  N O.       1   D E S I G N  R E S U L T S
M20                    Fe415 (Main)               Fe250 (Sec.)
LENGTH:  6400.0 mm      SIZE:   300.0 mm X  400.0 mm   COVER: 25.0 mm

DESIGN LOAD SUMMARY (KN MET)
----------------------------------------------------------------------------
SECTION |FLEXURE (Maxm. Sagging/Hogging moments) | SHEAR |           SHEAR |
         | P      MZ      MX      Load Case |     VY    MX    Load Case|
----------------------------------------------------------------------------
  0.0 | 0.00    0.00    0.00    0.00 | 0.00 | 0.00 | 1
  533.3 | 0.00  29.63  0.00 | 0.00 | 0.00 | 1
  1066.7 | 0.00  53.88  0.00 | 0.00 | 0.00 | 1
  1600.0 | 0.00  72.73  0.00 | 0.00 | 0.00 | 1
  2133.3 | 0.00  86.20  0.00 | 0.00 | 0.00 | 1
  2666.7 | 0.00  94.28  0.00 | 0.00 | 0.00 | 1
  3200.0 | 0.00  96.98  0.00 | 0.00 | 0.00 | 1
  3733.3 | 0.00  94.28  0.00 | 0.00 | 0.00 | 1
  4266.7 | 0.00  86.20  0.00 | 0.00 | 0.00 | 1
  4800.0 | 0.00  72.73  0.00 | 0.00 | 0.00 | 1
  5333.3 | 0.00  53.88  0.00 | 0.00 | 0.00 | 1
  5866.7 | 0.00  29.63  0.00 | 0.00 | 0.00 | 1
  6400.0 | 0.00    0.00    0.00    0.00 | 0.00 | 0.00 | 1

*** DESIGN SHEAR FORCE AT SECTION 0.0 IS 60.61 KN.
- CLAUSE 6.3.3 OF IS-13920
*** DESIGN SHEAR FORCE AT SECTION 6400.0 IS 60.61 KN.
- CLAUSE 6.3.3 OF IS-13920

NOTE:
MOMENT OF RESISTANCE IS CALCULATED BASED ON THE AREA OF STEEL PROVIDED.
IF AREA OF STEEL PROVIDED IS MUCH HIGHER COMPARED TO AREA OF STEEL
REQUIRED MOMENT OF RESISTANCE WILL INCREASE WHICH MAY INCREASE DESIGN
SHEAR FORCE.
------------------------------------------------------------------------
STAAD SPACE
-- PAGE NO.  7
```
D8.B.5 Column Design

Columns are designed for axial forces and biaxial moments per IS 456:2000. Columns are also designed for shear forces as per Clause 7.3.4. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS:456 have been taken care of in the column design of STAAD. However following clauses have been satisfied to incorporate provisions of IS 13920:

- The minimum grade of concrete shall preferably be M20. (Clause 5.2)
- Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3)
- The minimum dimension of column member shall not be less than 200 mm. For columns having unsupported length exceeding 4m, the shortest dimension of column shall not be less than 300 mm. (Clause 7.1.2)
- The ratio of the shortest cross-sectional dimension to the perpendicular dimension shall preferably be not less than 0.4. (Clause 7.1.3)
- The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided. (Clause 7.3.3)
- Special confining reinforcement shall be provided over a length $l_o$ from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length $l_o$ shall not be less than a) larger lateral dimension of the member at the section where yielding occurs, b) $1/6$ of clear span of the member, and c) 450 mm. (Clause 7.4.1)
- The spacing of hoops used as special confining reinforcement shall not exceed ¼ of minimum member dimension but need not be less than 75 mm nor more than 100 mm. (Clause 7.4.6)
- The area of cross-section of hoops provided are checked against the provisions for minimum area of cross-section of the bar forming rectangular, circular or spiral hoops, to be used as special confining reinforcement. (Clause 7.4.7 and 7.4.8)

D8.B.5.1 Column Design Output

Default column design output (TRACK 0.0) contains the reinforcement provided by STAAD and the capacity of the section. With the option TRACK 1.0, the output contains intermediate results such as the design forces, effective length coefficients, additional moments etc. All design output is given in SI units. An example of a column design output (with option TRACK 1.0) is given below.

<table>
<thead>
<tr>
<th>COLUMN NO.</th>
<th>DESIGN RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>M20</td>
<td></td>
</tr>
<tr>
<td>Fe415 (Main)</td>
<td></td>
</tr>
<tr>
<td>Fe415 (Sec.)</td>
<td></td>
</tr>
<tr>
<td>LENGTH: 3000.0 mm</td>
<td>CROSS SECTION:</td>
</tr>
<tr>
<td>350.0 mm X 400.0 mm</td>
<td>COVER: 40.0 mm</td>
</tr>
</tbody>
</table>
** GUIDING LOAD CASE:  5 END JOINT:
  2 SHORT COLUMN
DESIGN FORCES (KNS-MET)
------------------------
DESIGN AXIAL FORCE (Pu)
  :  226.7
About Z  
About Y
INITIAL MOMENTS
  :  0.64
   146.28
MOMENTS DUE TO MINIMUM ECC.
  :  4.53  
   4.53
SLENDERNESS RATIOS
  : -
- 
MOMENTS DUE TO SLENDERNESS EFFECT :
- 
- 
MOMENT REDUCTION FACTORS
  : -
- 
ADDITION MOMENTS (Maz and May)
  : -
- 
TOTAL DESIGN MOMENTS
  :  4.53
   146.28
** GUIDING LOAD CASE:  5
Along Z  Along Y
DESIGN SHEAR FORCES
  :  43.31
   76.08
REQD. STEEL AREA  :  3313.56  Sq.mm.
MAIN REINFORCEMENT : Provide 12 - 20 dia.
(2.69%,  3769.91  Sq.mm.)
(Equally distributed)
CONFINING REINFORCEMENT : Provide 10 mm dia.
  rectangular ties @ 85 mm c/c over a length 500.0 mm from each joint face towards midspan as per Cl. 7.4.6 of IS-13920.
TIE REINFORCEMENT
  : Provide 10 mm dia. rectangular ties @ 175 mm c/c
SECTION CAPACITY (KNS-MET)
--------------------------
Puz :  2261.52  Muz1 :
  178.71  Muy1 :  150.75
INTERACTION RATIO: 1.00 (as per Cl. 39.6, IS456:2000)
============================================================================
********************END OF COLUMN DESIGN RESULTS**************************
D8.B.6 Bar Combination

Initially the program selects only one bar to calculate the number of bars required and area of steel provided at each section along the length of the beam. You may use the BAR COMBINATION command to specify two bar diameters to calculate a combination of each bar to be provided at each section. The syntax for bar combination is given below.

START BAR COMBINATION

MD1 <bar diameter> MEMB <member list>

MD2 <bar diameter> MEMB <member list>

END BAR COMBINATION

Note: The bar sizes should be specified in the order of increasing size (i.e., MD2 bar diameter should be greater than MD1 bar diameter).

The beam length is divided into three parts, two at its ends and one at span. Ld gives the development length to be provided at the two ends of each section.

The typical output for bar combination is shown below:

<table>
<thead>
<tr>
<th>SECTION</th>
<th>0.0-1600.0</th>
<th>1600.0-4800.0</th>
<th>4800.0-6400.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOP</td>
<td>2-16í</td>
<td>2-16í</td>
<td>2-16í</td>
</tr>
<tr>
<td></td>
<td>in 1 layer(s)</td>
<td>in 1 layer(s)</td>
<td>in 1 layer(s)</td>
</tr>
<tr>
<td>Ast Reqd</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>Prov</td>
<td>402.29</td>
<td>402.29</td>
<td>402.29</td>
</tr>
<tr>
<td>Ld (mm)</td>
<td>752.2</td>
<td>1175.3</td>
<td>752.2</td>
</tr>
</tbody>
</table>

| BOTTOM   | 4-16í      | 2-16í + 2-25í | 4-16í        |
|          | in 1 layer(s) | in 1 layer(s) | in 1 layer(s) |
| Ast Reqd| 632.82     | 894.99       | 632.82       |
| Prov    | 804.57     | 1384.43      | 804.57       |
| Ld (mm) | 752.2      | 1175.3       | 752.2        |
D8.C. Indian Codes - Steel Design per IS 800 - 1984

STAAD.Pro is capable of performing steel design based on the Indian code IS 800 - 1984 General construction in steel - Code of practice.

**Note:** Steel design per the limit state method in IS 800 is also available in the Steel Design mode in the Graphical User Interface.

**Note:** The IS 800 - 1984 is not appropriate for the design of hollow pipe or tube sections. The design of such members should be done using IS 800 - 2007 WSD or LFD codes (on page 1942).

D8.C.1 Design Operations

STAAD.Pro contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.
- Specify design parameter values, if different from the default values.
- Specify whether to perform member selection by optimization.

These operations may be repeated by the user any number of times depending upon the design requirements. The entire ISI steel section table is supported. Section 11C.13 (on page 1928) describes the specification of steel sections.

D8.C.2 General Comments

This section presents some general statements regarding the implementation of Indian Standard code of practice (IS:800-1984) for structural steel design in STAAD.Pro. The design philosophy and procedural logistics for member selection and code checking are based upon the principles of allowable stress design.

Two major failure modes are recognized:

1. failure by overstressing
2. failure by stability considerations.

The following sections describe the salient features of the allowable stresses being calculated and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economic section is selected on the basis of least weight criteria. The code checking part of the program checks stability and strength requirements and reports the critical loading condition and the governing code criteria.

It is generally assumed that the engineer will take care of the detailing requirements like provision of stiffeners and check the local effects such as flange buckling and web crippling.
D8.C.3 Allowable Stresses

The member design and code checking in STAAD are based upon the allowable stress design method as per IS: 800 (1984). It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It would not be possible to describe every aspect of IS:800 in this manual. This section, however, will discuss the salient features of the allowable stresses specified by IS:800 and implemented in STAAD. Appropriate sections of IS:800 will be referenced during the discussion of various types of allowable stresses.

D8.C.3.1 Axial Stress

Tensile Stress

The allowable tensile stress, as calculated in STAAD as per IS:800 is described below.

The permissible stress in axial tension, \( \sigma_{at} \) in MPa on the net effective area of the sections shall not exceed

\[
\sigma_{at} = 0.6 \cdot f_y
\]

Where:

\( f_y \) = minimum yield stress of steel in Mpa

Compressive Stress

Allowable compressive stress on the gross section of axially loaded compression members shall not exceed 0.6\( f_y \) nor the permissible stress \( \sigma_{ac} \) calculated based on the following equation (per Clause: 5.1.1):

\[
\sigma_{ac} = 0.6\left\{ \left( \frac{f_{cc} \cdot f_y}{f_{cc} + (f_y)^n} \right)^{1/n} \right\}
\]

Where:

\( \sigma_{ac} \) = Permissible stress in axial compression, in MPa
\( f_y \) = Yield stress of steel, in Mpa
\( f_{cc} \) = Elastic critical stress in compression = \( \pi^2 \cdot E/\lambda^2 \)
\( E \) = Modulus of elasticity of steel, 2 X 10^5 Mpa
\( \lambda = l/r \) = Slenderness ratio of the member, ratio of the effective length to appropriate radius of gyration
\( n = A \) factor assumed as 1.4.

D8.C.3.2 Bending Stress

The allowable bending stress in a member subjected to bending is calculated based on the following formula: (Clause: 6.2.1)

\[
\sigma_{bt} \text{ or } \sigma_{bc} = 0.66 \cdot f_y
\]

Where:

\( \sigma_{bt} \) = Bending stress in tension
\( \sigma_{bc} \) = Bending stress in compression
\( f_y \) = Yield stress of steel, in MPa

For an I-beam or channel with equal flanges bent about the axis of maximum strength (z-z axis), the maximum bending compressive stress on the extreme fibre calculated on the effective section shall not exceed the values of maximum permissible bending compressive stress. The maximum permissible bending compressive stress shall be obtained by the following formula: (Clause: 6.2.2)
\[ o_{bc} = 0.66 \frac{f_{cb} f_y}{\left[(f_{cb})^n + (f_y)^n \right]^{1/n}} \]

Clause 6.2.3

Where:

- \( f_y \) = Yield stress of steel, in Mpa
- \( n \) = A factor assumed as 1.4.
- \( f_{cb} \) = Elastic critical stress in bending, calculated by the following formula:

\[ f_{cb} = k_1 \left( X + k_2 Y \right) \frac{c_2}{c_1} \]

Where:

\[ X = Y \sqrt{1 + \frac{1}{20} \frac{r_y^2}{D}} \text{ in MPa} \]
\[ Y = \frac{26.5(10)^5}{(1/r_y)^2} \]

- \( k_1 \) = a coefficient to allow for reduction in thickness or breadth of flanges between points of effective lateral restraint and depends on \( y \), the ratio of the total area of both flanges at the point of least bending moment to the corresponding area at the point of greatest bending moment between such points of restraint.
- \( k_2 \) = a coefficient to allow for the inequality of flanges, and depends on \( w \), the ratio of the moment of inertia of the compression flange alone to that of the sum of the moment of the flanges each calculated about its own axis parallel to the y-y axis of the girder, at the point of maximum bending moment.
- \( 1 \) = effective length of compression flange
- \( r_y \) = radius of gyration of the section about its axis of minimum strength (y-y axis)
- \( T \) = mean thickness of the compression flange, is equal to the area of horizontal portion of flange divided by width.
- \( D \) = overall depth of beam
- \( c_1, c_2 \) = respectively the lesser and greater distances from the section neutral axis to the extreme fibres.

**D8.C.3.3 Shear Stress**

Allowable shear stress calculations are based on Section 6.4 of IS:800. For shear on the web, the gross section taken into consideration consist of the product of the total depth and the web thickness. For shear parallel to the flanges, the gross section is taken as 2/3 times the total flange area.

**D8.C.3.4 Combined Stress**

Members subjected to both axial and bending stresses are proportioned accordingly to section 7 of IS:800. All members subject to bending and axial compression are required to satisfy the equation of Section 7.1.1.(a) for intermediate points, and equation of Section 7.1.1.(b) for support points.

For combined axial tension and bending the equation of Section 7.1.2. is required to be satisfied.

Cm coefficients are calculated according to the specifications of Section 7.1.3. Information regarding occurrence of sidesway can be provided through the use of parameters SSY and SSZ. In the absence of any user provided information, sidesway will be assumed.
D8.C.4 Design Parameters

In the STAAD.Pro implementation of IS:800, the user is allowed complete control of the design process through the use of design parameters. Available design parameters to be used in conjunction with IS:800 are listed in Table 7B.1 of this section along with their default values and applicable restrictions. Users should note that when the TRACK parameter is set to 1.0 and use in conjunction with this code, allowable bending stresses in compression (FCY & FCZ), tension (FTY & FTZ), and allowable shear stress (FV) will be printed out in Member Selection and Code Check output in Mpa. When TRACK is set to 2.0, detailed design output will be provided.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 167: Indian Steel Design IS 800:1984 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as INDIAN Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = design only for end moments and those at locations specified by the SECTION command. 1.0 = calculate section forces at twelfth points along the beam, design at each intermediate location and report the critical location where ratio is maximum.</td>
</tr>
<tr>
<td>CMY</td>
<td>0.85</td>
<td>Cm value in local y &amp; z axes</td>
</tr>
<tr>
<td>CMZ</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DFF</td>
<td>None (Mandatory for deflection check)</td>
<td>&quot;Deflection Length&quot; / Maxm. allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length&quot; (See Note 1)</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot; (See Note 1)</td>
</tr>
<tr>
<td>DMAX</td>
<td>100.0 cm.</td>
<td>Maximum allowable depth.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 cm.</td>
<td>Minimum allowable depth.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>FYLD</td>
<td>250 MPA (36.25 KSI)</td>
<td>Yield strength of steel.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis. Usually, this is minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis. Usually, this is major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length in local y-axis to calculate slenderness ratio.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in local z-axis (major).</td>
</tr>
<tr>
<td>MAIN</td>
<td>180 (Comp. Memb.)</td>
<td>Allowable Kl/r for slenderness calculations for compression members.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>PROFILE</td>
<td>-</td>
<td>Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2851) for details.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
</tr>
<tr>
<td>SSY</td>
<td>0.0</td>
<td>0.0 = Sidesway in local y-axis. 1.0 = No sidesway</td>
</tr>
<tr>
<td>SSZ</td>
<td>0.0</td>
<td>Same as above except in local z-axis.</td>
</tr>
<tr>
<td>TMAIN</td>
<td>400 (Tension Memb)</td>
<td>Allowable Kl/r for slenderness calculations for tension members.</td>
</tr>
</tbody>
</table>
### Parameter Name | Default Value | Description
---|---|---
**TRACK** | 0.0 | 0.0 = Suppress critical member stresses  
1.0 = Print all critical member stresses  
2.0 = Print expanded output. If there is deflection check it will also print the governing load case number for deflection check whenever critical condition for design is not DEFLECTION.(see Fig. 8B.1)

**UNF** | 1.0 | Same as above provided as a fraction of actual member length.

**UNL** | Member Length | Unsupported length for calculating allowable bending stress.

### Notes

**a.** "Deflection Length" is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the "Deflection Length" will be equal to the length of the member. However, in some situations, the "Deflection Length" may be different. A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.

![Diagram of beam with deflection](https://via.placeholder.com/150)

\[ D = \text{Maximum local deflection for members 1, 2, and 3.} \]

**PARAMETERS**

- **DFF 300 . ALL**
- **DJ1 1 ALL**
- **DJ2 4 ALL**

**b.** If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

**c.** The above parameters may be used in conjunction with other available parameters for steel design.
D8.C.5 Stability Requirements

Slenderness ratios are calculated for all members and checked against the appropriate maximum values. Section 3.7 of IS:800 summarizes the maximum slenderness ratios for different types of members. In STAAD implementation of IS:800, appropriate maximum slenderness ratio can be provided for each member. If no maximum slenderness ratio is provided, compression members will be checked against a maximum value of 180 and tension members will be checked against a maximum value of 400.

D8.C.6 Truss Members

A truss member is capable of carrying only axial forces. So in design no time is wasted in calculating bending or shear stresses, thus reducing design time considerably. Therefore, if there is any truss member in an analysis (like bracing or strut, etc.), it is wise to declare it as a truss member rather than as a regular frame member with both ends pinned.

D8.C.7 Deflection Check

This facility allows the user to consider deflection as a criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using three parameters which are described in D8.C.4 Design Parameters (on page 1919). Note that deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

D8.C.8 Code Checking

The purpose of code checking is to verify whether the specified section is capable of satisfying applicable design code requirements. The code checking is based on the IS:800 (1984) requirements. Forces and moments at specified sections of the members are utilized for the code checking calculations. Sections may be specified using the BEAM parameter or the SECTION command. If no sections are specified, the code checking is based on forces and moments at the member ends.

The code checking output labels the members as PASSED or FAILED. In addition, the critical condition (applicable IS:800 clause no.), governing load case, location (distance from the start) and magnitudes of the governing forces and moments are also printed out.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

D8.C.9 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, that is, the lightest section, which satisfies the applicable code requirements. The section selected will be of the same type (I-Section, Channel etc.) as originally specified by the user. Member selection may be performed with all types of steel sections listed in Section 11.C.12 (on page 1924) and user provided tables. Selection of members, whose properties are originally provided from user specified table, will be limited to sections in the user provided table. Member selection can not be performed on members whose cross sectional properties are specified as PRISMATIC.

The process of MEMBER SELECTION may be controlled using the parameters listed in D8.C.4 Design Parameters (on page 1919). It may be noted that the parameters DMAX and DMIN may be used to specify member depth constraints for selection. If PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one.
Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

D8.C.10 Member Selection By Optimization

Steel section selection of the entire structure may be optimized. The optimization method utilizes a state-of-the-art numerical technique which requires automatic multiple analysis. The user may start without a specifically designated section. However, the section profile type (BEAM, COLUMN, CHANNEL, ANGLE etc.) must be specified using the ASSIGN command (see Chapter 6). The optimization is based on member stiffness contributions and corresponding force distributions. An optimum member size is determined through successive analysis/design iterations. This method requires substantial computer time and hence should be used with caution.

Refer to TR.49.2 Member Selection by Optimization (on page 2854) for additional details.

D8.C.11 Tabulated Results of Steel Design

For code checking or member selection, the program produces the result in a tabulated fashion. The items in the output table are explained as follows:

| MEMBER | the member number for which the design is performed |
| TABLE  | the INDIAN steel section name which has been checked against the steel code or has been selected. |
| RESULT | prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark in front of the member number. |
| CRITICAL COND | the section of the IS:800 code which governs the design. |
| RATIO | prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed. |
| LOADING | provides the load case number which governs the design. |
| FX, MY, and MZ | provide the axial force, moment in local y-axis and moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX,MY and MZ are printed since they are the ones which are of interest, in most cases. |
| LOCATION | specifies the actual distance from the start of the member to the section where design forces govern. |

Note: If the parameter TRACK is set to 1.0, the program will block out part of the table and will print allowable bending stresses in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV). When the parameter TRACK is set to 2.0 for all members parameter code values are as shown in the following example.

```
STAAD.PRO CODE CHECKING - (IS-800) v1.0
*****************************************************
| Y | PROPERTIES |
| IN CM UNIT |
| MEM 7 | INDIAN SECTIONS |
| AX = 85.0 |
```
### D8.C.12 Indian Steel Table

This is an important feature of the program since the program will read section properties of a steel member directly from the latest ISI steel tables (as published in ISI-800). These properties are stored in memory corresponding to the section designation (e.g., ISMB250, etc.). If called for, the properties are also used for member design. Since the shear areas are built in to these tables, shear deformation is always considered for these members.

Almost all ISI steel tables are available for input. A complete listing of the sections available in the built-in steel section library may be obtained using the tools of the graphical user interface.
Following are the descriptions of all the types of sections available:

**D8.C.12.1 Rolled Steel Beams (ISJB, ISLB, ISMB and ISHB)**

All rolled steel beam sections are available the way they are designated in the ISI handbook (e.g., ISJB225, ISWB400, etc.)

<table>
<thead>
<tr>
<th>Section</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>20 TO 30 TA ST ISLB325</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** In case of two identical beams, the heavier beam is designated with an 'A' on the end (e.g., ISHB400 A, etc.).

<table>
<thead>
<tr>
<th>Section</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 5 TA ST ISHB400A</td>
<td></td>
</tr>
</tbody>
</table>

**D8.C.12.2 Rolled Steel Channels (ISJC, ISLC and ISMC)**

All these shapes are available as listed in ISI section handbook. Designation of the channels are per the scheme used by ISI.

<table>
<thead>
<tr>
<th>Section</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 TO 20 BY 2 TA ST ISMC125</td>
<td></td>
</tr>
<tr>
<td>12 TA ST ISLC300</td>
<td></td>
</tr>
</tbody>
</table>

**D8.C.12.3 Double Channels**

Back to back double channels, with or without spacing between them, are available. The letter D in front of the section name will specify a double channel (e.g., D ISJC125, D ISMC75, etc.).

<table>
<thead>
<tr>
<th>Section</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>21 22 24 TA D ISLC225</td>
<td></td>
</tr>
</tbody>
</table>

**D8.C.12.4 Rolled Steel Angles**

Both rolled steel equal angles and unequal angles are available for use in the STAAD implementation of ISI steel tables. The following example with explanations will be helpful in understanding the input procedure:

![Angle Symbol](image)

At present there is no standard way to define the local y and z axes for an angle section. The standard section has local axis system. The standard angle is specified as:

<table>
<thead>
<tr>
<th>Section</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>51 52 53 TA ST ISA60X60X6</td>
<td></td>
</tr>
</tbody>
</table>
This specification has the local z-axis (i.e., the minor axis corresponding to the V-V axis specified in the steel tables. Many engineers are familiar with a convention used by some other programs in which the local y-axis is the minor axis. STAAD provides for this convention by accepting the command:

54 55 56 TA RA ISA50X30X6

Tip: RA denotes reverse angle

**D8.C.12.5 Double Angles**

Short leg back-to-back or long leg back-to-back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle either LD or SD will serve the purpose. For example,

14 TO 20 TA LD ISA50X30X5 SP 1.5
23 27 TA SD ISA75X50X6

**D8.C.12.6 Rolled Tees (ISHT, ISST, ISLT and ISJT)**

All the rolled tee sections are available for input as they are specified in the ISI handbook. The following example illustrates the designated method.

1 2 5 8 TA ST ISNT100
67 68 TA ST ISST250

**D8.C.12.7 Pipes (Circular Hollow Sections)**

To designate circular hollow sections from ISI tables, use PIP followed by the numerical value of diameter and thickness of the section in mm omitting the decimal section of the value provided for diameter. The following example will illustrate the designation.

10 15 TA ST PIP 213.2

specifies a 213 mm dia. pipe with 3.2 mm wall thickness

Circular pipe sections can also be specified by providing the outside and inside diameters of the section. For example,

1 TO 9 TA ST PIPE OD 25.0 ID 20.0

specifies a pipe with outside dia. of 25 and inside dia. of 20 in current length units

Only code checking and no member selection will be performed if this type of specification is used.

**D8.C.12.8 Tubes (Rectangular or Square Hollow Sections)**

Designation of tubes from the ISI steel table is illustrated below.

For example,

15 TO 25 TA ST TUB 160808
Tubes, like pipes, can also be input by their dimensions (Height, Width and Thickness) and not by any table designs.

6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5

is a tube that has a height of 8, a width of 6, and a wall thickness of 0.5.

**Note:** Only code checking and no member selection is performed for TUBE sections specified this way.

**D8.C.12.9 Plate And Angle Girders (With Flange Plates)**

All plate and angle grinders (with flange plates) are available as listed in ISI section handbook. The following example with explanations will be helpful in understanding the input procedure.

A. Plate and angle girder symbol.
B. Web plate width in mm.
C. Web plate thickness in mm.
D. Flange angle, A X B X t, all in mm.

<table>
<thead>
<tr>
<th>Table 168: Flange angle key</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symbol</td>
</tr>
<tr>
<td>--------</td>
</tr>
<tr>
<td>A</td>
</tr>
<tr>
<td>B</td>
</tr>
<tr>
<td>C</td>
</tr>
<tr>
<td>E</td>
</tr>
</tbody>
</table>

E. Flange plate width in mm.
F. Flange plate thickness in mm.

**D8.C.12.10 Single Joist with Channels and Plates on the Flanges to be Used as Girders**

All single joist with channel and plates on the flanges to be used as girders are available as listed in ISI section handbook. The following example with explanations will be helpful in understanding the input procedure.
A. Joist Designation
   IW450 = ISWB450

B. Top flange channel designation:
   350 = ISMC350

C. Constant (always X).

D. Top flange plate thickness in mm.
   \textbf{Note:} D = 0 for no plate.

E. Bottom flange plate thickness in mm.
   \textbf{Note:} The heavier ISWB600 has been omitted, since the lighter ISWB600 is more efficient.

**D8.C.13 Column With Lacings And Battens**

For columns with large loads it is desirable to build rolled sections at a distance and inter-connect them. The joining of element sections is done by two ways:

a. Lacing
b. Batten

Double channel sections (back-to-back and face-to-face) can be joined either by lacing or by batten plates having riveted or welded connection.

Table 11C.3 gives the parameters that are required for Lacing or batten design. These parameters will have to be provided in unit NEW MMS along with parameters defined in **D8.C.4 Design Parameters** (on page 1919).

\textbf{Note:} Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.
Table 169: Parameters used in Indian Lacing or Batten steel member design.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CTYPE</td>
<td>1</td>
<td>Type of joining</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. implies single lacing with riveted connection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. implies double lacing with riveted connection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. implies single lacing with welded connection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4. implies double lacing with welded connection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5. implies batten with riveted connection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6. implies batten with welded connection</td>
</tr>
<tr>
<td>COG</td>
<td>0.0 mm</td>
<td>Center of gravity of the channel. This parameter is used when member properties are defined through user provided table using GENERAL option.</td>
</tr>
<tr>
<td>DBL</td>
<td>20 mm</td>
<td>Nominal diameter of rivet</td>
</tr>
<tr>
<td>DCFR</td>
<td>0.0</td>
<td>Used when member properties are defined through user provided table using GENERAL option.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. double channel back-to-back.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. double channel face-to-face.</td>
</tr>
<tr>
<td>EDIST</td>
<td>32 mm (Rivetted Connection) 25 mm (Welded Connection)</td>
<td>Edge Distance.</td>
</tr>
<tr>
<td>FVB</td>
<td>100 N/mm²</td>
<td>Allowable shear stress in rivet</td>
</tr>
<tr>
<td>FYB</td>
<td>300 N/mm²</td>
<td>Allowable bearing stress in rivet</td>
</tr>
<tr>
<td>SPA</td>
<td>0.0 mm</td>
<td>Spacing between double channels. This parameter is used when member properties are defined through user provided table using GENERAL option.</td>
</tr>
<tr>
<td>THETA</td>
<td>50 degree</td>
<td>Angle of inclination of lacing bars. It should lie between 40 degree and 70 degree.</td>
</tr>
</tbody>
</table>
D8.D. Indian Codes - Cold Formed Steel Design per IS 801 - 1975

STAAD.Pro is capable of performing steel design based on the Indian code IS 801 1975 *Code of practice for use of cold-formed light gauge steel structural members in general building construction*, reaffirmed in 2010. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations. Cold work of forming strengthening effects has been included as an option.

D8.D.1 Cross-Sectional Properties

You specify the geometry of the cross-section by selecting one of the section shape designations from the Gross Section Property Tables from IS:811-1987 (*Specification for cold formed light gauge structural steel sections*).

Checks per IS 801 are supported for the following profile shapes:

- Angle with lips
- Angle
- Channel with Lips
- Channel (or eave strut)
- Zee or zee purlin
- Zee with lips
- Hat
- Tube
- Pipe

The properties listed in the tables are gross section properties. STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

D8.D.2 Design Procedure

The design procedure according to IS 801 is based on an allowable stress method. In essence, the principle is to determine the maximum allowable design stress for a given section profile. The section properties that are used to calculate the allowable stress are based on the gross section properties or the “effective” section properties, based on the load being applied and the check being considered.

The design procedure varies slightly from profile to profile, based on the characteristics of the elements that form the profile shape. IS 801 classifies the elements (flat plates) that form the section profile into the following element types:

- stiffened element – an element that is supported along two opposite edges either by another element or a stiffening element such as a lip
- unstiffened element – an element that is supported by another element only along one edge
- multiple stiffened element – an element that is supported in between web(s) or stiffened element(s) by intermediate stiffeners that are parallel to the direction of stress

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WMIN</td>
<td>6 mm</td>
<td>Minimum thickness of weld</td>
</tr>
<tr>
<td>WSTR</td>
<td>108 N/mm²</td>
<td>Allowable welding stress</td>
</tr>
</tbody>
</table>
The various checks for a section profile and the design methods adopted depend on the type of the elements that form the profile. The following clauses from IS 801 are implemented and are used as appropriate:

- Cl 5.2.1
  - Cl.5.2.1.1
- Cl.5.2.3
- Cl.5.2.4
- Cl.6.1
  - Cl.6.1.1.1
- Cl.6.2
- Cl.6.3
- Cl.6.4
  - Cl.6.4.1
  - Cl.6.4.2
  - Cl.6.4.3
- Cl.6.6
  - Cl.6.6.1.1
  - Cl.6.6.1.2
  - Cl.6.6.1.3
- Cl.6.7
  - Cl.6.7.1
  - Cl.6.7.2
  - Cl.6.7.3
- Cl.6.8

**Note:** Deflection checks based on the effective section properties are not performed.

**D8.D.2.1 Maximum allowable section segment dimensions**

Checks will be done to verify the maximum allowable flat width ratios for the flange segments forming section profile as per Cl.5.2.3 and for the web segments as per Cl.5.2.4. Note that Cl.5.2.3(d) will not be considered as this can vary based on the amount of curling.

**D8.D.2.2 Maximum member slenderness**

Member slenderness checks will be performed for members subject to compression. The maximum allowable slenderness for a compression member will be limited to 200 as per Cl.6.3.3.

Slenderness for a member about both the z & y local axes will be calculated. If any of these exceeds the limiting value of 200, the program will consider that as a failure criterion. Note that members subject to tension will not have any slenderness checks performed.

**D8.D.2.3 Members subject to tension**

The following check will be performed for all members in tension:

\[
\text{Tensile stress ratio} = \frac{F_{\text{actual}}}{F}
\]

where
The maximum allowable design stress for members in tension will be calculated as:

\[ F = 0.6 \times F_y \]

where

\[ F_y = \text{the minimum yield strength of the section.} \]

If the increase in steel strength due to cold work forming is to be considered (ref. CWY param), then the design stress will be calculated as:

\[ F = 0.6 \times F_{ya} \]

where

\[ F_{ya} = \text{the average yield point of the full section and will be calculated as:} \]

\[ F_{ya} = C \times F_{yc} + (1 - C) F_{yt}. \]

( refer to Cl.6.6.1.a for the description of the terms used.)

Note: The FY and FU values are expected to be user inputs. The default value of FY will be taken as 3600 kg/cm²(Table 2 of IS:801) and FU will be taken as 4588 kg/cm².

D8.D.2.4 Members subject to compression

The following check will be performed for all members in compression:

\[ \text{Compression check ratio} = \frac{F_{\text{actual}}}{F_{\text{cap}}} \]

where

\[ F_{\text{actual}} = \text{the applied axial stress on the member} \]

\[ F_{\text{cap}} = \text{the allowable compressive stress for the member} \]

The compression capacity will be calculated as follows:

- For profile shapes with stiffened elements alone and not susceptible to Flexural torsional buckling (Tube):

  The maximum allowable design stress will be calculated as:

  \[ F = 0.6 \times F_y \]

  where

  \[ F_y = \text{the minimum yield strength of the section.} \]

Note: Allowance will be made for cold forming strength as required.

Note that if the effect of cold work forming is to be considered, then the enhanced strength \( F_{ya} \) will have to be determined based on the value of the factor Q. Hence to determine the factor Q, the program would use the design stress \( = 0.6 \times F_y \). This factor Q will then be used to determine the enhanced strength \( F_{ya} \), which will then be used for the design checks.

This design stress, \( F \), will be used to calculate the effective area of the section. The procedure to calculate the effective area involves calculating the effective width of each element that forms the profile. The effective width of each element will be calculated as described below:
• For elements without intermediate stiffeners:
  The effective width of such elements will be calculated as per the equations in Cl.5.2.1.1 (for closed square and rectangular tubes). The value of ‘F’ used will be the maximum allowable design stress $F$ as in section 3.2.1 above.

  **Note:** The current implementation does not allow for sections to have multiple stiffened elements as per Cl.5.2.1.2. Hence Cl.5.2.1.2 is not to be considered for calculating the effective width of elements.

  The maximum allowable compression stress for the member, $F_c$, will then be determined based on the equations as per Cl.6.6.1.1. Note that the factor $Q$ will be based on the effective area determined using the design stress $F$ as mentioned above.

• For profile shapes with stiffened elements alone (Pipe sections):
  The maximum allowable compressive stress for pipes will be determined as per cl.6.8:
  - For sections with $D/t < 232000 / F_y$, $F = 0.6 \times F_y$
  - For section with $232000 / F_y < D/t < 914000 / F_y$, $F = 46540 / (D/t) + 0.399 \times F_y$
  - Section with $D/t > 914000 / F_y$ will not be designed.
  A check will also be done to make sure that the compressive stress < $F_{a1}$ as determined by 6.6.1.1 for a value of $Q=1$.

• For profile shapes with un-stiffened elements alone (Angle sections):
  The maximum allowable design stress $F_c$ will be calculated as per the equations in Cl.6.2 (a), 6.2(b), 6.2(c) or 6.2 (d) as appropriate, based on the plate width-to-thickness ratio of each element.

  This design stress, $F_c$, will be used to calculate the factor $Q$ when checking for flexural/flexural torsional buckling failure modes. Note that since this section is subject to torsional buckling, the factor $Q$, to be used to calculate the allowable compressive stress $F_{a1}$ as per Cl. 6.6.1.2/3, will be calculated as the ratio between the $F_c$ for the element with the largest w/t ratio to the basic design stress as per Cl.6.1 & 6.2.

• For profile shapes with both stiffened and unstiffened elements and/or susceptible to Flexural torsional buckling (Channel, Zee, Hat, Channels with Lips, Zee with Lips, Angle with lip sections sections):
  These sections are subject to torsional-flexural buckling. The factor $Q$ will be determined as per Cl.6.6.1.1 (3) as:

  \[ Q = Q_s \times Q_a \]

  $Q_s$ will be calculated as the ratio between the $F_c$ for the element with the largest w/t ratio to the basic design stress as per Cl.6.1 & 6.2.

  $Q_a$ will be calculated for the stiffened elements as per Cl.6.6.1.1 (1) but with $F_a$ taken as the stress used to calculate $Q_s$.

  If $Q = 1.0$, the allowable stress $F_{a2}$ will be calculated as per Cl.6.6.1.2. If however the factor $Q < 1.0$, the allowable stress $F_{a2}$ will be calculated as per Cl.6.6.1.2, but with the term $Q$ being replaced with $Q \times F_y$.

**D.2.5 Members subject to bending**

The following check will be performed for all members in bending:

Bending check ratio = $M_{\text{actual}} / M_{\text{cap}}$

where

$M_{\text{actual}}$ = the applied bending stress on the member

$M_{\text{cap}}$ = the allowable bending stress of the member
This check will be done separately for bending about both the Z & Y axes.

The bending capacity of the section will be based on the maximum allowable bending stress, \( F_b \). The maximum allowable stress in a member subject to bending shall not exceed the following:

1. The maximum allowable stress for the extreme tension fiber as given in Cl.6.1

   \[
   F_{bt,\text{allowable}} = F = 0.6 \times F_y
   \]

   where

   \( F_y = \) the minimum yield strength of the section

2. The maximum allowable stress in the extreme compression fiber as given in Cl.6.2

   This will be calculated based on the \( w/t \) ratio of the compression element as per Cl.6.2(a), Cl.6.2(b), Cl.6.2(c), or Cl.6.2(d) as applicable. Note that these clauses will be applicable only for unstiffened elements in compression. For stiffened elements \( F_b \) will be taken as given in Cl.6.1

3. The maximum allowable lateral buckling stress as given in Cl.6.3 (a) or Cl.6.3 (b) as appropriate for the section profile.

   This clause will only be applicable for sections that are subject to bending about their major axis. For sections subject to minor axis bending this check will be ignored.

   The moment of inertia terms, \( I_{yc} \) and the section modulus term \( Z_{xc} \) in the equations will be based on the entire section. The value of the moment factor \( C_b \) will be based on the end moments of the analytical member being designed. If the member is subject to axial loads along with the bending moment, the value of \( C_b \) will be taken as 1.0

4. The maximum allowable bending stress in the webs of a given section (I, C, Hat, Tube, Z), \( F_{bw} \) shall be calculated as per Cl6.4.2 of the code.

   For pipe sections:

   The maximum allowable compressive stress for pipes will be determined as per cl.6.8:

   - For sections with \( D/t < 232000 / F_y \), \( F = 0.6 \times F_y \)
   - For sections with \( 232000 / F_y < D/t < 914000 / F_y \), \( F = 46540 / (D/t) + 0.399 F_y \)
   - Sections with \( D/t > 914000 / F_y \) will not be designed.

### D.2.6 Members subject to shear

The applied shear stress will be calculated based on the calculated shear area of the given section profile. The shear area for the various shapes will be taken as follows:

- Angle with Lips
  - For shear along the Y axis: \((D-t)\times t + 2.0 \times (\text{Lip Length} \times t)\)
  - For shear along the X axis: \((B-t)\times t\)

- Angle
  - For shear along the Y axis: \((D-t)\times t\)
  - For shear along the X axis: \((B-t)\times t\)

- Channel with Lips
  - For shear along the Y axis: \((D-2t)\times t + 2.0 \times (\text{Lip Length} \times t)\)
  - For shear along the X axis: \(2.0 \times B \times t\)

- Channel or Eave Strut
  - For shear along the Y axis: \((D-2t)\times t\)
For shear along the X axis: $2.0 \times B \times t$

- **Zee or Zee Purlin**
  - For shear along the Y axis: $(D-2t) \times t$
  - For shear along the X axis: $2.0 \times B \times t$

- **Zee with Lips**
  - For shear along the Y axis: $(D-2t) \times t + 2.0 \times (\text{Lip Length} - t) \times t$
  - For shear along the X axis: $2.0 \times B \times t$

- **Hat**
  - For shear along the Y axis: $2.0 \times (D-t) \times t$
  - For shear along the X axis: $B \times t + 2.0 \times (\text{Lip Length} - t) \times t$

- **Tube**
  - For shear along the Y axis: $2.0 \times (D-2t) \times t$
  - For shear along the X axis: $2.0 \times (B-2t) \times t$

- **Pipe**
  - For shear along the Y axis: $0.5 \times 2 \times \pi \times r \times t$
  - For shear along the X axis: $0.5 \times 2 \times \pi \times r \times t$

For shear checks along the local Y-axis, the maximum allowable shear stress, $F_v$, in the web of a section (I, C, Hat, Tube, Z) will be calculated as per Cl.6.4.1 based on the h/t ratio of the web element.

The code does not explicitly mention about shear checks along the horizontal (Z-axis). Hence, the program takes the maximum allowable shear stress along Z as $0.4 \times F_y$.

### D.2.7 Members subject to combined axial, bending, and shear

- Checks for members subject to combined axial compression and bending:
  
  The combined stress checks will be based on whether the member is susceptible to torsional buckling mode or not. The implementation will consider all loads as being applied through the shear center.

- **Members not susceptible to torsional buckling**
  
  For members that are not susceptible to torsional buckling, both the interaction equations as per Cl.6.7.1 will be checked as below:

  \[
  \frac{f_a}{F_{a1}} + \frac{(C_{mx} \times f_{bx})}{[(1- \frac{f_a}{F'_{ex}}) \times F_{bx}]} + \frac{(C_{my} \times f_{by})}{[(1- \frac{f_a}{F'_{ey}}) \times F_{by}]} \leq 1.0
  \]

  \[
  \frac{f_a}{F_{a0}} + \frac{f_{bx}}{F_{b1x}} + \frac{f_{by}}{F_{b1y}} \leq 1.0
  \]

  where

  \[
  f_a = \frac{P}{A}
  \]

  with $A$ = the full cross section area

  $f_{bx}$ = maximum applied bending stress about the x-axis. Note that $f_b$ for a section with stiffened compression element will be based on the effective width and the corresponding effective section modulus.

  $f_{by}$ = maximum applied bending stress about the y-axis. Note that $f_b$ for a section with stiffened compression element will be based on the effective width and the corresponding effective section modulus.
$C_m$ = a moment factor based on the ratio of the end moments and defined as $0.6 - 0.4 \times \frac{M_1}{M_2} \geq 0.4$. $M_1$ & $M_2$ are the smaller & larger end moments respectively, about the relevant axis.

$Fa_1$ = allowable compressive stress as per Cl.6.6.1.1

$F'e$ = $\frac{12 \times \pi^2 \times E}{23 \times (KL/r)^2}$

Note that $F'e$ will be calculated for the respective axes.

$Fa_0$ = is the allowable compressive stress from Cl.6.6.1.1 using $L = 0$.

$Fb_1$ = maximum bending stress without any lateral buckling as per Cl.6.1 and Cl.6.2 as appropriate

- Members susceptible to torsional buckling:

For members that are susceptible to torsional buckling, both the interaction equations as per Cl.6.7.2 (a) will be checked as below:

1. Sections that have the factor $Q = 1.0$:

The following two checks will be performed for all section profiles:

$$\frac{f_a}{F_{a1}} + \frac{f_b \times C_m}{F_{b1}[1 - \frac{f_a}{F'e}]} \leq 1.0 \text{ and } \frac{f_a}{F_{a0}} + \frac{f_b}{F_{b1}} \leq 1.0$$

where

$$fa = \frac{P}{A}$$

with $A$ = the full cross section area

$fb_1$ = maximum applied bending stress about the relevant axis. Note that $fb$ for a section with stiffened compression element will be based on the effective width and the corresponding effective section modulus.

$C_m$ = a moment factor based on the ratio of the end moments and defined as $0.6 - 0.4 \times \frac{M_1}{M_2} \geq 0.4$. $M_1$ & $M_2$ are the smaller & larger end moments respectively, about the relevant axis. Note that the end moments used will be the ones at the ends of the analytical member being designed.

$Fa_1$ = allowable compressive stress as per Cl.6.6.1.1

$F'e$ = $\frac{12 \times \pi^2 \times E}{23 \times (KL/r)^2}$

Note that $F'e$ will be calculated for the respective axes.

$Fa_0$ = the allowable compressive stress from Cl.6.6.1.1 using $L = 0$.

$Fb_1$ = maximum bending stress without any lateral buckling as per Cl.6.1 and Cl.6.2 as appropriate

Since the loads are being taken as being applied through the shear centre, the checks as per Cl.6.7.2 (b), Cl.6.7.2 (c) & Cl.6.7.2 (d) will not be performed.

2. Sections that have the factor $Q < 1.0$:

For members with a $Q$ factor less than 1.0, the interaction checks as per Cl.6.7.2 (a) will be performed, but with the term $F_y$ being replaced with $Q \times F_y$.

- Checks for members subject to combined shear and bending:

Only web elements of sections will be checked for the effects of combined bending & shear forces. The checks will be done as per Cl.6.4.3. The following check will be performed:
\[
\sqrt{\left(\frac{f_{bw}}{P_{bw}}\right)^2 + \left(\frac{f_v}{F_v}\right)^2} \leq 1.0
\]

where

\( f_{bw} = \) the applied bending stress at junction of flange & web
\( P_{bw} = \) \(\frac{36560000}{(h/t)^2}\) kgf/cm²
\( f_v = \) the applied shear stress
\( F_v = \) allowable shear stress as per Cl.6.4.1, but not limited to \(0.4 \times F_y\).

**D8.D.3 Code Checking and Member Selection**

The following two design modes are available:

**D8.D.3.1 Code Checking**

The program compares the resistance of members with the applied load effects, in accordance with IS:801-1975. Code checking is carried out for locations specified using the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. You may choose the degree of detail in the output data by setting the TRACK parameter.

Refer to [D1.B.1.3 Code Checking](on page 1418) for general information on Code Checking. Refer to [TR.49 Code Checking Specification](on page 2852) for details the specification of the Code Checking command.

**D8.D.3.2 Member Selection**

You may request that the program search the cold formed steel shapes database (standard sections) for an alternative section to pass the code check and meet the least weight criterion. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, presents design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

Refer to [D1.B.1.4 Member Selection](on page 1419) for general information on Member Selection. Refer to [TR.49.1 Member Selection Specification](on page 2853) for details the specification of the Member Selection command.

**D8.D.4 Design Parameters**

The following table contains the input parameters for specifying values of design variables and selection of design options.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as IS801 Design code to follow. See [TR.48.1 Parameter Specifications](on page 2851).</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>When this parameter is set to 0) the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (see TR.41 Section Specification (on page 2839) for details) For TRUSS members only start and end locations are designed. 1) the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member.</td>
</tr>
<tr>
<td>CB</td>
<td>0</td>
<td>Bending coefficient Cb used for bending checks. By default (value of 0), this is calculated by the program.</td>
</tr>
<tr>
<td>CMY</td>
<td>calculated</td>
<td>Coefficient Cm as per Cl. 6.7 for bending about the member Y axis. See IS:801-1975, 6.7. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.</td>
</tr>
<tr>
<td>CMZ</td>
<td>calculated</td>
<td>Coefficient Cm as per Cl. 6.7 for bending about the member Z axis. See IS:801-1975, 6.7. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.</td>
</tr>
<tr>
<td>CWY</td>
<td>0</td>
<td>Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See IS:801-1975, 6.1.1 0) effect should not be included 1) effect should be included</td>
</tr>
<tr>
<td>FLX</td>
<td>0</td>
<td>Specifies whether the member has torsional-flexural buckling restraints that will in turn determine whether the member is susceptible to flexural-torsional buckling mode. See IS:801-1975, 6.6.1 0) No Torsional restraints provided and the section subject to torsional flexural buckling 1) Torsional restraints provided and the section is not subject to torsional flexural buckling</td>
</tr>
<tr>
<td>FU</td>
<td>450 MPa (4588.72 kg/cm²)</td>
<td>Ultimate tensile strength of steel in current units.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>FYLD</td>
<td>353.04 MPa (\text{3600.0 kg/cm}^2)</td>
<td>Yield strength of steel in current units.</td>
</tr>
<tr>
<td>KX</td>
<td>1.0</td>
<td>Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor for overall buckling about the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor for overall buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LX</td>
<td>Member length</td>
<td>Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Effective length for overall buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Effective length for overall buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members. The net area is calculated as NSF × gross area.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual to allowable stresses</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are: 0) Prints only the member number, section name, ratio, and PASS/FAIL status. 1) Prints the design summary including allowable stress in addition to that printed by TRACK 0. 2) Prints member, material properties, and stress check table in addition to that printed by TRACK 1.</td>
</tr>
<tr>
<td>TSA</td>
<td>0</td>
<td>Specifies whether transverse web stiffeners have been provided to check for the requirements of IS: 801-1975, 5.2.4. 0) Web stiffeners not provided 1) Web stiffeners provided</td>
</tr>
<tr>
<td>UNL</td>
<td>Member length</td>
<td>Unsupported length (in current units) for calculating the allowable bending stress.</td>
</tr>
</tbody>
</table>

**D8.D.4 Design Results**

```
Track 2 Output

<table>
<thead>
<tr>
<th>Most critical result and corresponding forces.</th>
<th>MEMBER: 1 SECTION: G0CS40X4 LEN: 3.000 LOC: 0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>STATUS: FAIL</td>
<td>RATIO: 2.148</td>
</tr>
<tr>
<td>DESIGN FORCES:</td>
<td></td>
</tr>
<tr>
<td>Fx:(C) 0.000</td>
<td>Fy: 3.749</td>
</tr>
<tr>
<td>Mx: 0.000</td>
<td>My: 0.000</td>
</tr>
<tr>
<td>SECTION PROPERTIES: (Unit: CM)</td>
<td></td>
</tr>
<tr>
<td>Ag: 5.82000</td>
<td>Az: 3.20000</td>
</tr>
</tbody>
</table>
```
**Cz:** 1.75000  
**Cy:** 3.00000

**Iz:** 28.20000  
**Iy:** 12.30000  
**J:** 0.29600

**Sz:** 9.40000  
**Sy:** 5.46000

**Rz:** 2.20122  
**Ry:** 1.45375  
**Cw:** 162.00003

**MATERIAL INFO:**

(Unit: MPa)

**Fy:** 250.018  
**Fu:** 449.993  
**E:** 203404.356  
**G:** 77968.401

Cold-formed strength used for compression and bending checks respectively.

Fya(compression): 325.621  
Fya(bending): 387.521

**DESIGN PROPERTIES:**

**Member Length:** 3.000  
**Lz:** 3.000  
**Ly:** 3.000  
**Lb:** 3.000

Input parameters specified. Note Cb of 0 indicates that this value will be program-calculated.

**DESIGN PARAMETERS:**

**Kz:** 1.000  
**Ky:** 1.000  
**NSF:** 1.000  
**Cb:** 0.000

**CRITICAL SLENDERNESS:**

**Actual:** 0.000  
**Allowable:** 0.000  
**Ratio:** 0.000

**STAAD SPACE**

--- PAGE NO. 7

<table>
<thead>
<tr>
<th>CHECKS:</th>
<th>Stresses</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Loc.</td>
</tr>
<tr>
<td></td>
<td>(MET)</td>
</tr>
<tr>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>Tension</td>
<td>-</td>
</tr>
<tr>
<td>Compression</td>
<td>-</td>
</tr>
</tbody>
</table>

Bending stress checks at the critical compression and tension fibers for bending about Z.

| BendZComp | 0.000 | 2.25 | 1 | 239.014 | 232.512 | 1.028 | 6.3 |
| BendZTens | 0.000 | 2.25 | 1 | 239.014 | 232.512 | 1.028 | 6.3 |
| BendUnbraced | 0.000 | 2.25 | 1 | 239.014 | 111.292 | 2.148 | 6.3 LTB |
| BendYComp | - | - | - | - | - | 232.512 | - | 6.3 |
D8.E. Indian Codes - Steel Design per IS 800 - 2007

STAAD.Pro is capable of performing steel design based on the Indian code IS 800-2007 General construction in steel - Code of practice.

D8.E.1 General Comments

For steel design, STAAD.Pro compares the actual design forces with the capacities as defined by the Indian Standard Code. The IS 800: 2007 Code is used as the basis of this design.

A brief description of some of the major capacities is described herein.

The following commands should be used to initiate design per Limit State Method of this code:

\[
\text{PARAMETER } n \\
\text{CODE IS800 LSD}
\]

The following commands should be used to initiate design per Working Stress Method of this code:

\[
\text{PARAMETER } n \\
\text{CODE IS800 WSD}
\]

Where:

\[n = \text{optional integer (i.e., } -1, 2) \text{ which signifies the numerical order of parameter command block (if multiple blocks are specified).}\]

D8.E.2 Design Process

The design process follows the following design checks. When a design is performed, the output file reports the maximum utilization ratio from all of these checks.
**D8.E.2.1 Slenderness**

As per Section 3.8 Table 3, the slenderness ratio \((KL/r)\) of compression members shall not exceed 180, and the slenderness ratio \((L/r)\) of tension members shall not exceed 400.

You can edit the default values through MAIN and TMAIN parameters, as defined in [D8.E.4 Design Parameters](#) (on page 1951).

In version V8i (SELECTseries 6) (build 20.07.11.33) and later, the slenderness check is no longer used as a critical member ratio check. If a slenderness check is included and exceeds unity \((1.0)\), then it may be reported as the governing criteria. However, if it is less than unity, it will be reported but will not be used as the governing ratio.

**D8.E.2.2 Section Classification**

The IS 800: 2007 specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion.

Steel sections are classified as Plastic, Compact, Semi-Compact, or Slender element sections depending upon their local buckling characteristics.

This classification is a function of the geometric properties of the section as well as nature of the load applied to the member. The design procedures are different depending on the section class.

STAAD.Pro is capable of determining the section classification for the standard shapes and design the section for the critical load case accordingly. The Section Classification is done as per section 3.7 of IS 800:2007 and Table B2, for Outstanding and Internal Elements of a section.

For the criteria for being included in those classes, refer to section 3.7.2-(a) – (d) of the code.

Slender Sections

STAAD.Pro is capable of designing I-Sections with slender webs for IS 800:2007.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) or higher.

The IS:800-2007 code does not provide any clear guidelines about what method should be adopted for the design of slender section. The “Flange Only” methodology is used where it is assumed that flexure is taken by the flanges alone and the web will resist shear with adequate shear buckling resistance. This method requires that the flanges be non-slender elements (i.e., on the web is a slender element) to qualify for a valid section for design. If any of the flange elements become slender, the design will not be performed and a warning message is displayed in the output.

**D8.E.2.3 Tension**

Limit State Method

The criteria governing the capacity of Tension members are based on:

- Design Strength due to Yielding in Gross Section
- Design Strength due to Rupture of Critical Section
- Design Strength due to Block Shear

STAAD.Pro calculates the tension capacity of a given member based on these three limit states.

The limit state of yielding in the gross section is intended to prevent excessive elongation of the member, and the corresponding check is done as per section 6.2 of the code.
The Design strength, involving rupture at the section with the net effective area, is evaluated as per section 6.3 of the code. Here, the number of bolts in the connection may be specified through the use of the design parameter \( \text{ALPHA} \).

The Design strength, involving block shear at an end connection, is evaluated as per section 6.4 of the code. This criteria is made optional by the parameter \( \text{DBS} \). If the value of \( \text{DBS} \) is specified as 1, additional design parameters \( \text{AVG}, \text{AVN}, \text{ATG}, \text{and ATN} \) must be supplied to the program for that member.

The Net Section Area may be specified through the use of the parameter \( \text{NSF} \).

Working Stress Method

The criteria governing the allowable stress from tension in members are based on Section 11.2.1 of the code:

- Yielding of Gross Section - to prevent excessive elongation of the member due to material yielding.
- Rupture of Net Section - to prevent rupture of the net effective section area. The number of bolts in the connection may be specified through the use of the design parameter \( \text{ALPHA} \). The code parameter, \( \gamma_{\text{M1}} \), is taken as 1.25 per Table 5, Clause 5.4.1 of the code.
- Block Shear — to prevent block shearing at the end connection. This check is made option through use of the \( \text{DBS} \) parameter. Additional design parameters \( \text{AVG}, \text{AVN}, \text{ATG}, \text{and ATN} \) must be supplied to the program for any member which is to be checked for block shear. The code parameters, \( \gamma_{\text{M0}} \) and \( \gamma_{\text{M1}} \), are taken as 1.10 and 1.25, respectively, per Table 5, Clause 5.4.1 of the code.

**Note:** Block shear is *not* checked by default.

These criteria are dependent on the steel material yield stress parameter, \( \text{FYLD} \), and ultimate tensile strength parameter, \( \text{FU} \).

### D8.E.2.4 Compression

The design capacity of the section against Compressive Force, the guiding phenomenon is the flexural buckling.

**Limit State Method**

The buckling strength of the member is affected by residual stress, initial bow and accidental eccentricities of load.

To account for all these factors, the strength of the members subjected to axial compression is defined by buckling class a, b, c or d as per clause 7.1.2.2 and Table 7 of IS 800:2007.

Imperfection factor, obtained from buckling class, and Euler’s Buckling Stress ultimately govern compressive force capacity of the section as per clause 7.1.2 of IS 800:2007.

**Working Stress Method**

The actual compressive stress is given by:

\[
f_c = \frac{F_X}{A_e}
\]

where:

\( A_e \) = The effective section area as per Clause 7.3.2 of the code. This is equal to the gross cross sectional area, \( AX \), for any non-slower (plastic, compact, or semi-compact) section class (on page 1943). In the case of slender sections, this is limited to value of \( A_e \) as described below.

The permissive compressive stress is calculated by first determining the Buckling Class of the section per Table 10 of the code and \( \alpha_{YY} \) & \( \alpha_{ZZ} \) based on Table 7.
Design

D. Design Codes

\[ F_{ac} = 0.6 \cdot F_{cd} \]

where:

\[ F_{cd} = \text{the minimum of the values of } F_{cd} \text{ calculated for the local Y and Z axis.} \]

\[ F_{cd} = (Y_{mo})/ \left[ \varphi + (\varphi^2 + \lambda^2) \right] \]

\( \lambda \) = the non-dimensional slenderness factor is evaluated for each local Y and Z axis.

\[ \lambda = (F_{cc})^{1/2} \]

\[ \varphi = 0.5 \left[ 1 + a(\lambda - 0.2) + \lambda^2 \right] \]

\( F_{cc} \) = the Euler Buckling Stress.

\[ F_{cc} = \pi^2 \cdot E/(Kl/r)^2 \]

\( K \) = the effective length factor for bending about either the local Y or Z axis, as provided in the \( KY \) and \( KZ \) parameters, respectively.

\( r \) = radius of gyration about the local Y or Z axis for the section.

\( FYLD \) = The yield strength of steel specified in the FYLD parameter.

Slender Sections

For member with slender section under axial compression, design compressive strength should be calculated on area ignoring depth thickness ratio of web in excess of the class 3 (semi-compact) limit.

Refer to clause 7.3.2 and Table 2 of IS 800:2007, (corresponding to “Internal Element of Compression Flange”)

\[ A_e = A_g - (d/t_w - 42\varepsilon) \cdot t_w^2 \]

where:

\( A_e \) = Effective area of section.

\( A_g \) = Gross area of section.

\( d \) = Depth of web.

\( t_w \) = thickness of web.

Flexural-Torsional Buckling for Single Angles

In the case of single angles (ST and RA), an axial compression load does not pass through the member centroid. In practical applications, the load must pass through one of the legs. The deviation between the load and the centroid can be considerable, and thus the effect of flexural-torsional buckling must be considered.

The parameters of \( ANG, FXTY, \) and \( NBL \) are used to control these calculations.

The calculations for the flexural-torsional buckling strength of a single angle member in compression are performed per section 7.5.1.2 of the code (either limit state method or working stress method are applicable).

**D8.E.2.5 Shear**

The design capacities of the section against Shear Force in major- and minor-axis directions are evaluated as per section 8.4 of the code, taking care of the following phenomena:

- Nominal Plastic Shear Resistance
- Resistance to Shear Buckling
Shear area of the sections are calculated as per sec. 8.4.1.1.
Nominal plastic shear resistance is calculated as per sec. 8.4.1.
Among shear buckling design methods, Simple post-critical method is adopted as per sec. 8.4.2.2(a).

Working Stress Design
The actual shear stress is determined about the major and minor axes, respectively:

\[ \tau_{bY} = \frac{F_Y}{A_Y} \]
\[ \tau_{bZ} = \frac{F_Z}{A_Z} \]

The permissible shear stress is determined as:

a. When subjected to pure shear:
\[ \tau_{ab} = 0.40 \cdot F_Y \]
b. When subjected to shear buckling:
\[ \tau_{ab} = 0.70 \cdot V_n \cdot A_v \]

where
\[ V_n = \text{Nominal Shear Strength as per Clause 8.4.2.2.(a)} \]
\[ = V_{cr} = \tau_b \cdot A_v \]
\[ A_v = A_Y \text{ or } A_Z, \text{ whichever is appropriate, with reference to Clause 8.4.1.1.} \]

Shear buckling must be checked when \((d/t_w) > 67 \cdot \epsilon_w\) for webs without stiffener or \((d/t_w) > 67 \cdot \epsilon_w \cdot \sqrt{(K_v/5.35)}\) for webs with stiffeners.

where
\[ d = \text{Clear Depth of Web between Flanges.} \]
\[ t_w = \text{Thickness of Web.} \]
\[ \epsilon_w = \sqrt{\left( \frac{250}{F_y} \right)} \]
\[ F_y = \text{Yield Strength of Web, specified using the FYLD parameter.} \]
\[ K_v = \text{Shear Buckling Coefficient:} \]
\[ = 5.35, \text{ when transverse stiffeners are provided only at supports.} \]
\[ = 4.0 + 5.35 / (c/d)^2 \text{ for } (c/d) < 1.0 \]
\[ = 5.35 + 4.0 / (c/d)^2 \text{ for } (c/d) \geq 1.0 \]
\[ c = \text{Spacing of Transverse Stiffeners} \]
\[ \mu = \text{Poisson’s Ratio} \]
\[ \tau_b = \text{Shear Stress corresponding to Web-buckling:} \]
\[ = \frac{F_y}{\sqrt{3}}, \text{ when, } \lambda_w \leq 0.8 \]
\[ = (1 - 0.8 \cdot (\lambda_w \cdot 0.8)) \cdot (F_y / \sqrt{3}) \text{ when, } 0.8 < \lambda_w < 1.2 \]
\[ = F_y / (\sqrt{3} \cdot \lambda_w^2) \text{ when, } \lambda_w \geq 1.2 \]
\[ \tau_{cr,e} = \text{The Elastic Critical Shear Stress of the Web} \]
\[ \tau_{cr,e} = (K_v \cdot n^2 \cdot E) / (12 \cdot (1 - \mu^2) \cdot (d/t_w)^2) \]

where
\[ \lambda_w = \text{Non-dimensional Web Slenderness Ratio for Shear Buckling Stress.} \]
\[ \lambda_w = \left[ F_y / (\sqrt{3} \cdot \tau_{cr,e}) \right]^{1/2} \]

Slender Sections
Slender sections should be verified against shear buckling resistance if $d/t_w > 67 \cdot \varepsilon$ for web without stiffeners or if it exceeds $67 \cdot \varepsilon \cdot \sqrt{(K_v/5.35)}$ for a web with stiffeners.

Design methods for resistance to shear buckling are described in clause 8.4.2.2 of IS:800-2007 code.

$$V_n = V_{cr}$$

where

$$V_{cr} = \text{shear force corresponding to web buckling}$$

$$\tau_b = \text{shear stress corresponding to web buckling, determined as follows:}$$

i. When $\lambda_w \leq 0.8$

$$\tau_b = \frac{f_{yw}}{\sqrt{3}}$$

ii. When $0.8 < \lambda_w < 1.2$

$$\tau_b = [1 - 0.8(\lambda_w - 0.8)]\left(\frac{f_{yw}}{\sqrt{3}}\right)$$

iii. When $\lambda_w \geq 1.2$

$$\tau_b = \frac{f_{yw}}{(\sqrt{3} \lambda_w^2)}$$

$$\lambda_w = \text{non-dimensional web slenderness ratio or shear buckling stress, given by:}$$

$$= \left[ f_{yw}\left(\sqrt{3} \tau_{cr,e}\right) \right]^{1/2}$$

$$\tau_{cr,e} = \text{elastic critical shear stress of the web}$$

$$= \frac{k_v \pi^2 E}{12 \cdot (1 - \mu^2) \cdot (d/t_w)^2}$$

$$\mu = \text{Poisson's ratio}$$

$$K_v = \text{5.35 when transverse stiffeners are provided only at supports}$$

$$= 4.0 + 5.35/(c/d)^2 \text{ for } c/d < 1.0$$

$$= 5.35 + 4.0/(c/d)^2 \text{ for } c/d \geq 1.0$$

$$c = \text{spacing of transverse stiffeners}$$

$$d = \text{depth of the web}$$

**D8.E.2.6 Bending**

The design bending moment capacity of a section is primarily dependent on whether the member is laterally supported or unsupported.

You can control the lateral support condition of the member by the use of LAT parameter.

If the member is laterally supported, then the design strength is calculated as per the provisions of the section 8.2.1 of IS 800:2007, based on the following factors:

- Whether section with webs susceptible to shear buckling before yielding
- Shear Force to Design Shear Strength Ratio
- Section Classification

If the member is laterally unsupported, then the design strength is calculated as per the provisions of the section 8.2.2 of IS 800:2007, based on the following factors:

- Lateral Torsional Buckling
- Section Classification
- Working Stress Design
Actual bending stress values are given by, about major (Z) and minor (Y) axes, respectively:

\[ f_{bcz} = \frac{M_z}{Z_{ecz}} \]
\[ f_{btz} = \frac{M_z}{Z_{etz}} \]
\[ f_{bcy} = \frac{M_y}{Z_{ecy}} \]
\[ f_{bty} = \frac{M_y}{Z_{ety}} \]

The permissible bending stress is given as follows:

**a. For laterally supported beams:**

\[ F_{abc} = F_{abt} = 0.66 \cdot F_y \] for Plastic or Compact sections
\[ F_{abc} = F_{abt} = 0.60 \cdot F_y \] for Semi-compact sections

where \( F_y \) = Yield strength of steel, indicated by the FYLD parameter.

**b. For laterally unsupported beams:**

i. About the major axis:

\[ f_{abcz} = 0.60 \cdot \frac{M_d}{Z_{ecz}} \]
\[ f_{abtz} = 0.60 \cdot \frac{M_d}{Z_{etz}} \]

where \( M_d \) = Design Bending Strength as per Clause 8.2.2

\[ = \beta_b \cdot Z_{pz} \cdot f_{bd} \]
\[ f_{bd} = \chi_{LT} \cdot F_y / \gamma_{mo} \]
\[ Z_{ez} = \text{Elastic Section Modulus of the Section} \]
\[ Z_{pz} = \text{Plastic Section Modulus of the Section} \]
\[ \alpha_{LT} = \begin{cases} 0.21 & \text{for Rolled Steel Section and 0.49 for Welded Steel Section} \\ 1.0 & \text{for Plastic and Compact Section or } Z_{ez}/Z_{pz} \text{ for Semi-Compact Section} \end{cases} \]
\[ \beta_b = \begin{cases} 1.0 & \text{for Plastic and Compact Section or } Z_{ez}/Z_{pz} \text{ for Semi-Compact Section} \\ \frac{Z_{ez}}{Z_{pz}} \end{cases} \]
\[ \lambda_{LT} = \text{Non-dimensional slenderness ratio} \]
\[ \lambda_{LT} = (\beta_b \cdot Z_{pz} \cdot F_y / M_{cr})^{1/2} \leq (1.2 \cdot Z_{ez} \cdot F_y / M_{cr})^{1/2} \]
\[ \phi_{LT} = 0.5 \cdot (1 + \alpha_{LT} \cdot (\lambda_{LT} - 0.2) + \lambda_{LT}^2) \]
\[ \chi_{LTZ} = \text{The Bending Stress Reduction Factor to account for Lateral Torsional Buckling} \]
\[ \chi_{LTZ} = \frac{1}{\phi_{LTZ} + \sqrt{\phi_{LTZ}^2 - \lambda_{LTZ}^2}} \]
\[ Z_{ecz} = \text{Elastic Section Modulus of the section about Major Axis for the compression side} \]
\[ Z_{etz} = \text{Elastic Section Modulus of the section about Major Axis for the tension side} \]
\[ M_{cr} = \sqrt{\frac{n^2 E I_y}{L^2_{LT}}} \left( G I_t + \frac{n^2 E I_w}{L^2_{LT}} \right) \]
\[ I_y = \text{Moment of inertia about the minor axis} \]
\[ L_{LT} = \text{Effective length for lateral torsional buckling as determined using either} \]
\[ \text{the } KX \text{ or } LX \text{ parameters} \]
\[ I_t = \text{Torsional constant of the section} \]
\[ I_t = \text{Warping constant of the section} \]
\[ G = \text{Shear modulus of the material} \]
ii. About the minor axis, the permissible bending stress is calculated as for a laterally supported section.

Slender Sections

For member with slender section subjected to bending, moment is taken by flanges alone. Design bending strength should be calculated with effective elastic modulus disregarding the contribution of web of the section.

\[
Z_{ex} = 2 \left[ B_f \cdot t_f^3/12 + (B_f \cdot t_f) \cdot (D/2 - t_f/2)^2 \right] / (0.5 \cdot D)
\]

\[
Z_{ey} = 2 \left( B_f \cdot t_f^3/12 \right) / (0.5 \cdot B_f)
\]

Where:

- \( Z_{ex} \) = Elastic Section modulus about major principal axis
- \( Z_{ey} \) = Elastic Section modulus about minor principal axis
- \( B_f \) = Width of flange
- \( t_f \) = thickness of flange
- \( D \) = Overall depth of section

The Moment Capacity will be \( M_d = Z_e \cdot f_y / \gamma_{m0} \) for “Laterally Supported” condition.

The Moment Capacity will be \( M_d = Z_e \cdot f_{bd} / \gamma_{m0} \) for “Laterally Un-Supported” condition.

Where, \( f_{bd} \) is defined in clause 8.2.2 of IS:800-2007 (described in previous Working Stress Design section).

**Note:** Slender section can only attain elastic moment capacity and cannot reach to plastic moment capacity.

### D8.E.2.7 Combined Interaction Check

Members subjected to various forces – axial, shear, moment, torsion - are checked against combined interaction check.

**Limit State Method**

This interaction check is done taking care of two aspects:

- Section Strength
- Overall Member Strength

Section Strength interaction ratio is calculated as per sec. 9.3.1 of the code.

Overall Member Strength interaction ratio is calculated as per sec. 9.3.2, taking care of the design parameters PSI, CMX, CMY and CMZ.

**Working Stress Method**

The following interactions are considered:

a. Combined Bending and Shear — No reduction in allowable stresses for the interaction of bending and shear is considered.

b. Combined Axial Compression and Bending — The following formulas are intended to require member stability:

\[
f_c/f_{acy} + 0.6 \cdot K_y (C_{my} f_{bcy}/f_{acy}) + K_L T f_{bcy}/f_{abcy} \leq 1.0
\]

\[
f_c/f_{acz} + 0.6 \cdot K_y (C_{my} f_{bcy}/f_{acy}) + K_x f_{bcz}/f_{abcz} \leq 1.0
\]

\[
f_c/(0.6 f_y) + f_{bcy}/f_{acy} + f_{bcz}/f_{abcz} \leq 1.0
\]

Where:
Actual axial compressive stress.

Allowable compressive stress, governed by buckling, about the local Y and Z axis, respectively.

Actual bending compressive stress about minor and major axes, respectively.

Allowable bending compressive stress about minor and major axes, respectively.

\[
K_y = 1 + (\lambda_y \cdot 0.2) \cdot n_y \leq 1 + 0.8 \cdot n_y
\]

\[
K_z = 1 + (\lambda_z \cdot 0.2) \cdot n_z \leq 1 + 0.8 \cdot n_z
\]

\[
K_{LT} = 1 - 0.1 \cdot \lambda_{LT} \cdot n_y/(C_{mLT} \cdot 0.25) \geq 0.1 \cdot n_y/(C_{mLT} \cdot 0.25)
\]

c. Combined Axial Tension and Bending — The following formulas are intended to require member stability:

\[
f_t/f_{at} + f_{bt}\cdot f_{abty} + f_{btz}/f_{abtz} \leq 1.0
\]

Where:

- \( f_t \) = Actual axial tensile stress.

- \( f_{at} \) = Allowable axial tensile stress.

- \( f_{bt}\cdot f_{abty} \) = Actual bending tensile stress about minor and major axes, respectively.

- \( f_{btz}/f_{abtz} \) = Allowable bending tensile stress about minor and major axes, respectively.

### D8.E.2.8 Minimum Web Thickness

The minimum web thickness is checked against the serviceability requirements in 8.6.1.1 and the compression flange buckling requirement of 8.6.1.2.

If these sections are not satisfied, then a warning message is included in the output, including the clause which did not meet the web thickness requirements.

This check applies to the following sections:

- Sections with a stiffened web (i.e., connected to flanges along both longitudinal edges):
  - wide flange (rolled, UPT, tapered)
  - I section with cover plates
  - double I-section side-by-side
  - channel
  - double channel (back-to-back and front-to-front)
  - tube, HSS rectangle

- Sections with an unstiffened web (i.e., connected to flange along only one longitudinal edge):
  - tee
  - single angle
  - double angle
  - star angle

### D8.E.3 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD.Pro may be used (namely: I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, or Double Channel section).

Member properties may also be specified using the User Table facility except for the General and Prismatic member types.
**D8.E.3.1 Star Angle Arrangements**

STAAD.Pro can design "star angle" sections (double angles, toe to toe) per IS 800:2007. Members using this section must be axial only (i.e., use TRUSS specification). It is assumed that the star angle arrangement is a welded shape. Plated shapes are not accounted for in the program.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 4) or higher.

The internal cross section properties are calculated for the principal axes and are checked for Tension and Compression limit states as described in this section.

**D8.E.4 Design Parameters**

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

**Table 171: Indian Steel Design IS 800:2007 Parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as IS800 LSD Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
</tbody>
</table>
| ALPHA          | 0.8           | A Factor, based on the end-connection type, controlling the Rupture Strength of the Net Section, as per Section 6.3.3:  
  0.6 = For one or two bolts  
  0.7 = For three bolts  
  0.8 = For four or more bolts |
| ANG            | 0             | The location of the angle section through which the axial load passes by. It will only work if the section is single angle. The options are:  
  0 = Centroid.  
  1 = any of the legs.  
  If the value is zero, it means the member is loaded through centroid. In that case Flexural-torsional buckling will not be checked. |
<p>| ATG            | None (Mandatory for Block Shear check) | Minimum Gross Area in Tension from the bolt hole to the toe of the angle, end bolt line, perpendicular to the line of the force. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ATN</td>
<td>None (Mandatory for Block Shear check)</td>
<td>Minimum Net Area in Tension from the bolt hole to the toe of the angle, end bolt line, perpendicular to the line of the force.</td>
</tr>
<tr>
<td>AVG</td>
<td>None (Mandatory for Block Shear check)</td>
<td>Minimum Gross Area in shear along bolt line parallel to external force.</td>
</tr>
<tr>
<td>AVN</td>
<td>None (Mandatory for Block Shear check)</td>
<td>Minimum Net Area in shear along bolt line parallel to external force.</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>0.0 = design at ends and those locations specified by the SECTION command. 1.0 = design at ends and at every 1/12th point along member length (default).</td>
</tr>
<tr>
<td>CAN</td>
<td>0.0</td>
<td>Beam Type, used for both deflection checks and the limit of bending strength in clause 8.2.1.2: 0 = General Member, no limit used in clause 8.2.1.2, member deflection taken as the distance from the displaced end points of the member (D</td>
</tr>
<tr>
<td>CMX</td>
<td>0.9</td>
<td>Equivalent uniform moment factor for Lateral Torsional Buckling(as per Table 18, section 9.3.2.2)</td>
</tr>
<tr>
<td>CMY</td>
<td>0.9</td>
<td>Cm value in local Y &amp; Z axes, as per Section 9.3.2.2.</td>
</tr>
<tr>
<td>CMZ</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DFF</td>
<td>None(Mandatory for deflection check)</td>
<td>“Deflection Length” / Maximum allowable local deflection.</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of “Deflection Length”.</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of “Deflection Length”.</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000 in.</td>
<td>Maximum allowable depth.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 in.</td>
<td>Minimum allowable depth.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>FU</td>
<td>420 MPA</td>
<td>Ultimate Tensile Strength of Steel in current units.</td>
</tr>
<tr>
<td>FXTY</td>
<td>0</td>
<td>The fixity of the connection between gusset plate and each end of member. Any value between 0 and 1 is allowed where 1 = Fixed and 0 = Hinged. Default is hinged. A value between 0 and 1 means the level of fixity that is between hinged and fixed. This value will be used to calculate the coefficients $k_1$, $k_2$ and $k_3$ of table 12 by linear interpolation between their specified values for the two extreme (hinged and fixed) conditions.</td>
</tr>
<tr>
<td>FYLD</td>
<td>250 MPA</td>
<td>Yield Strength of Steel in current units.</td>
</tr>
<tr>
<td>KX</td>
<td>1.0</td>
<td>Effective Length Factor for Lateral Torsional Buckling (as per Table-15, Section 8.3.1)</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local Y-axis. Usually, the Minor Axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local Z-axis. Usually, the Major Axis.</td>
</tr>
</tbody>
</table>
| LAT            | 0.0           | Specifies lateral support of beam, as per Section 8.2.1 and 8.2.2, respectively:  
0 = Beam is laterally unsupported  
1 = Beam is laterally supported |
| LST            | 0             | Defines the number of longitudinal stiffeners used:  
0 = No longitudinal stiffener  
1 = Longitudinal stiffener is provided at 0.2D of web from the compression flange  
2 = Longitudinal stiffeners are provided at 0.2D and 0.5D of the web from the compression flange |
| LX             | Member Length | Effective Length for Lateral Torsional Buckling (as per Table-15, Section 8.3.1) |
| LY             | Member Length | Length to calculate Slenderness Ratio for buckling about local Y axis. |
| LZ             | Member Length | Same as above except in Z-axis (Major). |
| MAIN           | 180           | Allowable Slenderness Limit for Compression Member (as per Section 3.8)  
0 = the default value will be used  
-1 = the slenderness check will not be performed (any other negative value will be ignored |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| NBL            | 1             | The number of bolts connecting each end of member with the gusset plate. The options are:  
|                |               | 0 = one bolt.  
|                |               | 1 = two or more bolts or weld. |
| NSF            | 1.0           | Net Section Factor for Tension Member. |
| TMAIN          | 400           | Allowable Slenderness Limit for Tension Member (as per Section 3.8)  
|                |               | 0 = the default value will be used  
|                |               | -1 = the slenderness check will not be performed (any other negative value will be ignored |
| PROFILE        | None          | Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2851) for details. |
| PSI            | 1.0           | Ratio of the Moments at the ends of the laterally unsupported length of the beam, as per Section 9.3.2.1:  
|                |               | 0.8 = where Factored Applied Moment and Tension can vary independently  
|                |               | 1.0 = For any other case |
| RATIO          | 1.0           | Permissible ratio of the actual to allowable stresses. |
| STP            | 1             | Specifies the section type per Table 2 and Table 10:  
|                |               | 1 = Hot rolled section  
|                |               | 2 = Welded section |
| TRACK          | 0             | Controls the levels of detail to which results are reported.  
|                |               | 0 = Minimum detail  
|                |               | 1 = Intermediate detail level  
|                |               | 2 = Maximum detail |
| TSP            | 0             | Spacing of transverse stiffeners. |
| TST            | 0             | Used to control transverse stiffeners in design:  
|                |               | 0 = No Transverse Stiffener is provided  
|                |               | 1 = Transverse Stiffener is provided |
D8.E.5 Code Checking and Member Selection

Both Code Checking and Member Selection options are available for the IS 800: 2007 code.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

D8.E.5.1 Example 1

Commands for code checking

UNIT NEWTON METER
PARAMETER 1
CODE IS800 LSD
ALPHA 0.7 ALL
DBS 1 ALL
CAN 1 MEMB 2
PSI 0.8 MEMB 2
TMAIN 350 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2

D8.E.5.2 Example 2

Commands for member selection

UNIT NEWTON METER
PARAMETER 1
CODE IS800 LSD
MAIN 160 MEMB 7
KY 0.8 MEMB 7
KZ 0.9 MEMB 7
FYLD 350 ALL
SELECT ALL

D8.E.6 Amendment 1 - January 2012

The various sections of Amendment-1 to the IS:800-2007 Code, dated January 2012, are implemented in STAAD.Pro as applicable.

Please refer the link below for the details of the list and their implementation status in STAAD.Pro.


D8.F. Indian Codes - Concrete Design per IS 13920-2016

STAAD.Pro is capable of performing concrete design based on the 2016 edition of the Indian code IS 13920 Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces - Code of Practice. Designs per IS 13920-2016 satisfy all provisions of IS 456-2000 and IS 13920-2016 for beams and columns (Refer to D8.A. Indian Codes - Concrete Design per IS 456 (on page 1887)).
The IS:13920 code provides the design and detailing requirements of reinforced concrete (RC) members to resist lateral effects of earthquake shaking. This gives the RC members sufficient strength and ductility to resist severe earthquake shaking without collapse.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
• RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10 (on page 94)

D8.F.1 Scope
The following reinforced concrete members that can be designed by the latest revision are the following:

a. Reinforced concrete beams (excluding deep beams)
b. Reinforced concrete columns (see note b (on page 1956) for details)

The following types of cross sections for concrete members can be designed:

• For beams:
  prismatic (rectangular and square)
  T-beams
• For columns:
  prismatic: rectangular, square, and circular

Note: IS13920 does not deal with plate design. Plate design is to be performed per IS:456:2000.

Notes
a. The clauses that are already implemented for IS-13920:1993 revision and remains unchanged in revision 2016 are not discussed in these design notes.
b. Implementation of Clause 7.2 of IS:13920-2016 is excluded. This clause checks the relative strength of beams and columns at a joint and finds out if any column at a joint is acting as a gravity column or is a part of lateral load resisting system. STAAD.Pro does not have any gravity load column as a structural element. It considers all columns as a lateral load resisting column. Hence this check is currently excluded.
c. Implementation of Clause 9.0 of IS:13920-2016 for design of beam-column joint for distortional shear is currently excluded.
d. The provisions for the spacing of for special confining reinforcement per Amendment 1 to IS:13920-2016 are not implemented. The design of this reinforcement is designed to the initial IS:13920-2016 edition.

D8.F.2 Beam Design

Beams are designed for flexure, shear, and torsion. If required the effect of the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The total number of sections considered is 13. All of these sections are scanned to determine the design force envelopes.

For design to be performed as per IS:13920 2016 the width of the member shall not be less than 200 mm (Clause 6.1.2). Also the member shall preferably have a width-to-depth ratio of more than 0.3 (Clause 6.1.1).

The factored axial stress on the member should not exceed $0.08f_{ck}$ (Clause 6.1) for all active load cases. If it exceeds allowable axial stress no design will be performed.
Cl 6.1 of IS:13920-2016 is applicable for beams resisting earthquake induced effects. The program performs a load type check where it determines if the design load contains earthquake or dynamic load or not. If yes, it will perform check against Cl 6.1. Otherwise, the program will ignore the design load for check against this code provision.

**D8.F.2.1 Design for Flexure**

Design procedure is same as that for IS 456. However while designing following criteria are satisfied as per IS13920 2016:

1. The minimum grade of concrete shall preferably be M20. (Clause 5.2)
2. Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3.1)
3. The minimum tension steel ratio on any face, at any section, is given by (Clause 6.2.1b)
   \[
   \rho_{\text{min}} = \frac{0.24f_{ck}}{f_y}
   \]
   The maximum steel ratio on any face, at any section, is given by (Clause 6.2.2)
   \[
   \rho_{\text{max}} = 0.025
   \]
4. The longitudinal steel on bottom at a joint face must be at least equal to half the steel at its top at the same section. (Clause 6.2.3)
5. The steel provided at each of the top and bottom face, at any section, shall at least be equal to one-fourth of the maximum negative moment steel provided at the face of either joint. (Clause 6.2.4)
6. Beams shall have at least two 12 mm diameter bars each at the top and bottom faces.

**D8.F.2.2 Design for Shear**

The shear force to be resisted by vertical hoops is guided by the Clause 6.3.3 of IS 13920:2016 revision. Elastic sagging and hogging moments of resistance of the beam section at ends are considered while calculating shear force. Plastic sagging and hogging moments of resistance can also be considered for shear design if the PLASTIC parameter is used (refer to D8.F.4 Design Parameters (on page 1958)).

Shear reinforcement is calculated to resist both shear forces and torsional moments. The procedure is the same as that used for IS 456.

The following criteria are satisfied while performing design for shear as per Cl. 6.3.5 of IS-13920 2016:

- The spacing of vertical hoops over a length of 2d at either end of the beam shall not exceed
  a. $d/4$
  b. 6 times the diameter of the smallest longitudinal bars
  c. 100 mm

  The spacing calculated from above, if less than that calculated from IS 456 considerations, is provided.

- While calculating shear force reinforcement the provision of Clause 6.3.4c (i.e. neglecting design shear capacity of concrete of RC section) is taken into consideration. The spacing of links is provided in accordance with clause 6.3.5.2 of IS-13920 2016.

**D8.F.3 Column Design**

Columns are designed for axial forces and biaxial moments per IS 456:2000. Columns are also designed for shear forces as per Clause 7.3.4. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS:456 are performed by the column design of STAAD.Pro. However following clauses have been satisfied to incorporate provisions of IS 13920 2016:
• The minimum grade of concrete shall preferably be M20. (Clause 5.2)
• Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3.1)
• The minimum dimension of column member shall not be less than:
  a. 20 times the diameter of the largest longitudinal reinforcement in the beam passing through or anchoring into the column at the joint, or
  b. 300 mm (Clause 7.1.1)
• The ratio of the shortest cross-sectional dimension to the perpendicular dimension shall preferably be not less than 0.45. (Clause 7.1.2)
• The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided. (Clause 7.4.2d)
• The minimum diameter of the rectangular hoop is 8 mm when diameter of the longitudinal bar is less than or equal to 32 mm, and 10 mm when diameter of longitudinal bar is more than 32 mm. (Clause 7.4.2a)
• Special confining reinforcement shall be provided over a length $l_o$ from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length $l_o$ shall not be less than:
  a. larger lateral dimension of the member at the section where yielding occurs,
  b. 1/6 of clear span of the member,
  c. 450 mm (Clause 8.1a)
• The spacing of hoops used as special confining reinforcement shall not exceed:
  a. ¼ of minimum member dimension,
  b. 6 times diameter of the smallest longitudinal bars, or
  c. 100 mm (Clause 8.1b)
• The area of cross-section of hoops provided are checked against the provisions for minimum area of cross-section of the bar forming rectangular, circular, or spiral hoops, to be used as special confining reinforcement. (Clause 8.1c)

CI 7.1 of IS:13920-2016 is applicable for columns resisting earthquake induced effects. The program will check whether the design load is gravity load case (as defined by the GLD parameter) or earthquake load. If yes, the program will perform this code check. Otherwise, the program will ignore the design load case for check against this code provision.

**Note:** In previous releases, the design performed checks against this clause for *all* types of design load cases, irrespective of whether they contain earthquake load or not. This logic is now automated to include only earthquake or gravity load case.

**D8.F.4 Design Parameters**

The program contains a number of parameters that are needed to perform design as per IS 13920 2016. It accepts all parameters that are needed to perform design as per IS:456. Over and above it has some other parameters that are required only when designed is performed as per IS:13920 2016. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. The following table contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.
### Table 172: Indian Concrete Design IS 13920 2016 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CODE</strong></td>
<td>-</td>
<td>Must be specified as IS13920 2016 or ISH13920 Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2859).</td>
</tr>
</tbody>
</table>
| **BRACING**   | 0.0           | Beam Design  
1.0 = the effect of axial force will be taken into account for beam design.  
Column Design: Correspond to the terms "Braced" and "Unbraced" described in Notes 1, 2, and 3 of Clause 39.7.1 of IS456:2000.  
1.0 = the column is unbraced about major axis.  
2.0 = the column is unbraced about minor axis.  
3.0 = the column is unbraced about both axis. |
| **DEPTH**     | YD            | Total depth to be used for design. This value defaults to YD (depth of section in Y direction) as provided under MEMBER PROPERTIES. |
| **CLB**       | 25 mm         | Clear cover to stirrups at bottom for beams.  
**Note:** If both CLEAR and CLB are defined for beams, CLB is considered in design. Concrete cover to main reinforcement at bottom of beam will be calculated from CLB. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| CLEAR          | >30 mm 40 mm  | Concrete cover to main reinforcement bars for beams  
Concrete cover to main longitudinal bars for columns  
**Notes:**  
1. This is the clear cover to the outermost main reinforcing bar. It is not the clear cover for the stirrups or the tie bars.  
2. If the CLB parameter is not defined, then the CLEAR parameter will be used by default. |
| CLT            | 25 mm         | Clear cover to stirrups at top for beams.  
**Note:** If both CLEAR and CLT are defined for beams, CLT is considered in design. Concrete cover to main reinforcement at top of beam will be calculated from CLT. |
| COMBINE        | 0.0           | Default value means there will be no member combination.  
1.0 = no printout of sectional force and critical load for combined member in the output.  
2.0 = printout of sectional force for combined member in the output.  
3.0 = printout of both sectional force and critical load for combined member in the output. ***  
**Note:** Refer description of COMBINE parameter at the end of the parameter table. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EFACE</td>
<td>0.0</td>
<td>Face of support location at end of beam. The parameter can also be used to check against shear at any point from the end of the member.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Both SFACE and EFACE are input as positive numbers. Not valid along with COMBINE parameter. *</td>
</tr>
<tr>
<td>ELY</td>
<td>1.0</td>
<td>Ratio of effective length to actual length of column about minor axis.</td>
</tr>
<tr>
<td>ELZ</td>
<td>1.0</td>
<td>Ratio of effective length to actual length of column about major axis.</td>
</tr>
<tr>
<td>ENSH</td>
<td>0.0</td>
<td>Perform shear check against enhanced shear strength as per Cl. 40.5 of IS456:2000.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0 = ordinary shear check to be performed (no enhancement of shear strength at sections close to support)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>a positive value (say x) = shear strength will be enhanced up to a distance x from the start of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note after Table 8A.1)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>a negative value (say –y) = shear strength will be enhanced up to a distance y from the end of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note after Table 8A.1)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0 = the program will calculate Length to Overall Depth ratio. If this ratio is greater than 2.5, shear strength will be enhanced at sections (&lt;2d) close to support otherwise ordinary shear check will be performed.</td>
</tr>
</tbody>
</table>
## Design

### D. Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EUDL</td>
<td>None</td>
<td>Equivalent u.d.l on span of the beam. This load value must be the unfactored load on span. During design the load value is multiplied by a factor 1.2. If no u.d.l is defined factored shear force due to gravity load on span will be taken as zero. No elastic or plastic moment will be calculated. Shear design will be performed based on analysis result. (Refer note)</td>
</tr>
<tr>
<td>FYMAIN</td>
<td>415 N/mm²</td>
<td>Yield Stress for main reinforcing steel.</td>
</tr>
<tr>
<td>FYSEC</td>
<td>415 N/mm²</td>
<td>Yield Stress for secondary reinforcing steel.</td>
</tr>
<tr>
<td>FC</td>
<td>Fcu or 30 N/mm²</td>
<td>Concrete Yield Stress. The FC value will be taken from Material Definition where Compressive Strength (Fcu) is defined. If the input is not available, the design will consider 30 MPa as default value.</td>
</tr>
</tbody>
</table>
### Design

#### D. Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| GLD            | None          | Gravity load number to be considered for calculating equivalent u.d.l on span of the beam, in case no EUDL is mentioned in the input. This load case can be any static load case containing **MEMBER LOAD** on the beam which includes **UNI**, **CON**, **LIN** and **TRAP** member loading. **CMOM** member loading is considered only when it is specified in local direction. **FLOOR LOAD** is also considered.

The load can be primary or combination load. For combination load only load numbers included in load combination is considered.

The load factors are ignored. Internally the unfactored load is multiplied by a factor 1.2 during design.

If both EUDL and GLD parameters are mentioned in the input mentioned EUDL will be considered in design.

**Note:** No dynamic (Response spectrum, 1893, Time History) and moving load cases are considered.

**CMOM** member loading in global direction is not considered.

**UMOM** member loading is not considered. |
| HLINK          | Spacing of longitudinal bars measured to the outer face | Longer dimension of the rectangular confining hoop measured to its outer face. It shall not exceed 300 mm as per Cl. 7.4.8.

If the HLINK value as provided in the input file does not satisfy the clause the value will be internally assumed as the default one. This parameter is valid for rectangular column. |
### Design

**D. Design Codes**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| IPLM           | 0.0           | Default value calculates elastic/plastic hogging and sagging moments of resistance of beam at its ends.  
1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at start node of beam. This implies no support exists at start node.  
-1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at start node of beam. This implies support exists at start node.  
2.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at end node of beam. This implies no support exists at end node.  
-2.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at end node of beam. This implies support exists at end node. **  

**Note:** Not valid along with COMBINE parameter. **
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| IMB            | 0.0           | Default value calculates elastic/plastic hogging and sagging moments of resistance of beam at its ends.  
1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at both ends of beam. This implies no support exist at either end of the member.  
-1.0 = calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at both ends of beam. This implies support exist at both ends of the member.** |
| MINMAIN        | 10 mm         | Minimum main reinforcement bar size. |
| MAXMAIN        | 60 mm         | Maximum main reinforcement bar size. |
| MINSEC         | 8 mm          | Minimum secondary reinforcement bar size. |
| MAXSEC         | 12 mm         | Maximum secondary reinforcement bar size. |
| PLASTIC        | 0.0           | Default value calculates elastic hogging and sagging moments of resistance of beam at its ends.  
1.0 = plastic hogging and sagging moments of resistance of beam to be calculated at its ends. |
| RATIO          | 4.0           | Maximum percentage of longitudinal reinforcement in columns. |

**Note: Not valid along with COMBINE parameter.**
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| REINF          | 0.0           | 0.0 = Tied column (default)  
1.0 = spiral reinforcement |
| RENSH          | 0.0           | Distance of the start or end point of the member from its nearest support. This parameter is used only when a span of a beam is subdivided into two or more parts. Refer to Notes (on page 1894) for IS456 Design Parameters. |
| RFACE          | 4.0           | 4.0 = longitudinal reinforcement in column is arranged equally along four faces.  
2.0 invokes two faced distribution about major axis.  
3.0 invokes two faced distribution about minor axis. |
| SFACE          | 0.0           | Face of support location at start of beam. It is used to check against shear at the face of the support in beam design. The parameter can also be used to check against shear at any point from the start of the member.*  
*Note: Both SFACE and EFACE are input as positive numbers.* |
| SPSMAIN        | 25 mm         | Minimum clear distance between main reinforcing bars in beam and column. For column center to center distance between main bars cannot exceed 300 mm. |
| TORISON        | 0.0           | 0.0 = torsion to be considered in beam design.  
1.0 = torsion to be neglected in beam design. |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| TRACK          | 0.0           | Beam Design:  
0) output consists of reinforcement details at START, MIDDLE and END.  
1) critical moments are printed in addition to TRACK 0 output.  
2) required steel for intermediate sections defined by NSECTION are printed in addition to TRACK 1 output.  
Column Design:  
0) reinforcement details are printed.  
1) column interaction analysis results are printed in addition to TRACK 0 output.  
2) a schematic interaction diagram and intermediate interaction values are printed in addition to TRACK 1 output.  
9) details of section capacity about major and minor axes are printed in addition to TRACK 1 output |
| ULY            | 1.0           | Ratio of unsupported length to actual length of column about minor axis. |
| ULZ            | 1.0           | Ratio of unsupported length to actual length of column about major axis. |
| WIDTH          | ZD            | Width to be used for design. This value defaults to ZD as provided under MEMBER PROPERTIES. |

Bar combination has been introduced for detailing. Please refer [D8.6 Bar Combination](on page 1915) for details.

* EFACE and SFACE command is not valid for member combination.

** IPLM and IMB commands are not valid for member combination. These commands are ignored for members forming physical member.

*** The purpose of COMBINE command is the following:
1. If a beam spanning between two supports is subdivided into many sub-beams this parameter will combine them into one member. It can also be used to combine members to form one continuous beam spanning over more than two supports.

2. When two or more members are combined during design plastic or elastic moments will be calculated at the column supports. At all the intermediate nodes (if any) this calculation will be ignored.

Note: Please note that the program only recognizes column at right angle to the beam. Inclined column support is ignored.

3. It will calculate sectional forces at 13 sections along the length of the combined member.

4. It will calculate critical loads (similar to that of Design Load Summary) for all active load cases during design. Beams will be combined only when DESIGN BEAM command is issued.

The following lines should be satisfied during combination of members:

1. Members to be combined should have same sectional properties if any single span between two column supports of a continuous beam is subdivided into several members.

2. Members to be combined should have same constants (E, Poi ratio, alpha, density, and beta angle)

3. Members to be combined should lie in one straight line.

4. Members to be combined should be continuous.

5. Vertical members (i.e., columns) cannot be combined.

6. Same member cannot be used more than once to form two different combined members.

7. The maximum number of members that can be combined into one member is 299.

Note: Sectional forces and critical load for combined member output will only be available when all the members combined are successfully designed in both flexure and shear.

ENS and RENS parameters will have to be provided (as and when necessary) even if physical member has been formed.

**D8.F.3.1 Example**

The following lines show a standard example for design to be performed in IS 13920 2016.

```
STAAD SPACE
UNIT METER MTON
JOINT COORDINATES
... ... ...
MEMBER INCIDENCES
... ... ...
MEMBER PROPERTY INDIAN
... ... ...
CONSTANTS
... ... ...
SUPPORTS
... ... ...
DEFINE 1893 LOAD
ZONE 0.05 I 1 K 1 B 1
SELFWEIGHT
JOINT WEIGHT
... ... ...
LOAD 1 SEISMIC LOAD IN X DIR
1893 LOAD X 1
LOAD 2 SEISMIC LOAD IN Z DIR
1893 LOAD Z 1
LOAD 3 DL
```
MEMBER LOAD
...... UNI GY -5
LOAD 4 LL
MEMBER LOAD
...... UNI GY -3
LOAD COMB 5 1.5(DL+LL)
 3 1.5 4 1.5
LOAD COMB 6 1.2(DL+LL+SLX)
 1 1.2 3 1.2 4 1.2
LOAD COMB 7 1.2(DL+LL-SLX)
 1 1.2 3 1.2 4 -1.2
LOAD COMB 8 1.2(DL+LL+SLZ)
 2 1.2 3 1.2 4 1.2
LOAD COMB 9 1.2(DL+LL-SLZ)
 2 1.2 3 1.2 4 -1.2
PDELTA ANALYSIS
LOAD LIST 5 TO 9
START CONCRETE DESIGN
CODE IS13920 2016
UNIT MMS NEWTON
FYMAIN 415 ALL
FC 20 ALL
MINMAIN 12 ALL
MAXMAIN 25 ALL
TRACK 2.0 ALL
*** Unfactored gravity load on members 110 to 112 is 8 t/m (DL+LL) i.e. 78.46 N/mm
EUDL 78.46 MEMB 110 TO 112
** Members to be combined into one physical member
COMBINE 3.0 MEMB 110 TO 112
*** Plastic moment considered
PLASTIC 1.0 MEMB 110 TO 112
DESIGN BEAM 110 TO 112
DESIGN COLUMN ........
END CONCRETE DESIGN
FINISH

D9. Japanese Codes

D9.A. Japanese Codes - Concrete Design per 1991 AIJ

STAAD.Pro is capable of performing concrete design based on the Japan code AIJ 1991 *Architectural Institute of Japan Standards for Structural Calculation of Steel Reinforced Concrete Structures*. Design for a member involves calculation of the amount of reinforcement required for the member. Calculations are based on the user specified properties and the member forces obtained from the analysis. In addition, the details regarding placement of the reinforcement on the cross section are also reported in the output.

Design of members per AIJ requires the *STAAD Japan Design Codes* SELECT Code Pack.

D9.A.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed:
• For Beams — Prismatic (Rectangular and Square)
• For Columns — Prismatic (Rectangular, Square, and Circular)

D9.A.2 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the MEMBER PROPERTY command. These are the \( D \) (YD) and \( b \) (ZD) dimensions for rectangular or square cross sections and the \( D \) (YD) for circular cross sections.

The following is an example the required input:

```
UNIT MM
MEMBER PROPERTY
  1 3 TO 7 9 PRISM YD 450. ZD 250.
  11 13 PR YD 350.
```

In the above input, the first set of members are rectangular (450 mm depth and 250 mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with a 350 mm diameter.

**Caution:** It is absolutely imperative that you do not provide the cross section area (AX) as an input.

D9.A.3 Slenderness Effects and Analysis Considerations

Slenderness effects are extremely important in designing compression members. Slenderness effects result in additional forces being exerted on the column over and above those obtained from the elastic analysis. There are two options by which the slenderness effects can be accommodated.

The first option is to compute the secondary moments through an exact analysis. Secondary moments are caused by the interaction of the axial loads and the relative end displacements of a member. The axial loads and joint displacements are first determined from an elastic stiffness analysis and the secondary moments are then evaluated.

The second option is to approximately magnify the moments from the elastic analysis and design the column for the magnified moment. It is assumed that the magnified moment is equivalent to the total moment comprised of the sum of primary and secondary moments.

STAAD provides facilities to design according to both of the above methods. To utilize the first method, the command `PDELTA ANALYSIS` must be used instead of the `PERFORM ANALYSIS` in the input file. The user must note that to take advantage of this analysis, all the combinations of loading must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a primary load case is revised during the P-delta analysis based on the deflections. Also, note that the proper factored loads (like 1.5 for dead load etc.) should be provided by the user. STAAD does not factor the loads automatically. The second method mentioned above is utilized by providing the magnification factor as a concrete design parameter (See the parameter `MMAG` in D9.A.7 Design Parameters (on page 1973)). The column is designed for the axial load and total of primary and secondary biaxial moments if the first method is used and for the axial load and magnified biaxial moments if the second method is used.

D9.A.4 Beam Design

Beams are designed for flexure, shear, and torsion. The program considers 12 equally spaced divisions of the beam member. However this number can be redefined by the `NSECTION` parameter. All these sections are designed for flexure, shear and torsion for all load cases. The results include design results for most critical load case.

**Design Codes**

Example
UNIT KG CM
START CONCRETE DESIGN
CODE JAPAN
FYMAIN SRR295 ALL
FYSEC SRR295 ALL
FC 350 ALL
CLEAR 2.5 MEM 2 TO 6
TRACK 1.0 MEMB 2 TO 9
DESIGN BEAM 2 TO 9
END CONCRETE DESIGN

D9.A.4.1 Design for Flexure

Reinforcement for positive and negative moments are calculated on the basis of section properties provided by the user. Program first try to design the section for g = 0 and pt = balanced reinforcement ratio. If allowable moment is lower than the actual moment program increases g value for same pt and checks the satisfactory conditions. If conditions are not satisfied this procedure continues until g reaches to 1.0 and then pt value is increased keeping g = 1.0. This procedure continues until pt reaches to its maximum value( 2 % ). But if the allowable moment for pt = maximum value and g = 1.0 is lower than the actual moment the program gives message that the section fails.

This program automatically calculates the Bar size and no. of bars needed to design the section. It arranges the bar in layers as per the requirements and recalculate the effective depth and redesign the sections for this effective depth.

Notes
a. Beams are designed for MZ only. The moment MY is not considered in flexure design
b. MMAG parameter can be used to increase design moment
c. 1.4 cm. is added to the clear cover to take stirrup size into consideration for flexure design.
d. STAAD beam design procedure is based on the local practice and considering the fact that Japan is a high seismic zone area.

D9.A.4.2 Design for Shear

The Design Shear value, Q_d, is evaluated for the beam. The update effective depth is used to then calculate the allowable shear stress. The allowable shear stress of concrete, f_s, is automatically calculated from design load type (permanent or temporary) and given density of concrete. The program then calculates the required bar size, a_w, and spacing of stirrups. The reinforcement ratio for the stirrup, p_w, is calculated for design Bar size and stirrup pitch and all the necessary checking is done.

For seismic loading it is needed to increase shear force ≥ 1.5 times the actual value and this can be done utilizing the Design Shear Modification factor, k (SMAG parameter) without changing the Design Moment.

Notes:

a. Stirrups are always assumed to be 2-legged
b. Governing density to determine Light weight or Normal Weight Concrete is 2.3 kg/sq. cm
D9.A.4.3 Design for Torsion

Torsion design for beam is optional. If the TORSION parameter value is 1.0, the program will design the assigned beam(s) for torsion. The program first checks whether extra reinforcement is needed for torsion or not. If additional reinforcement is needed, this additional pt is added to flexure pt and additional Pw is added to shear design Pw.

D9.A.5 Column Design

Columns are designed for axial force, MZ moment, MY moment, and shear force. Both the ends of the members are designed for all the load cases and the loading which produces largest amount of reinforcement is called as critical load. If Track 0 or Track 1 is used, design results will be printed for critical load only. But if Track 2 is used, you can get detailed design results of that member. The value of Pt needed for minimum axial force, maximum axial force, maximum MZ, maximum MY among all the load cases for both the ends will be printed. If the MMAG parameter is used, the column moments will be multiplied by that value. If the SMAG parameter is used, column shear force will be multiplied by that value.

Column design is done for Rectangular, Square and Circular sections. For rectangular and square sections Pt value is calculated separately for MZ and MY, while for circular sections Pg value is calculated for MZ and MY separately.

Column design for biaxial moments is optional. If the BIAXIAL parameter value is 1.0, the program will design the column for biaxial moments. Otherwise column design is always uniaxial.

Steps involved:

1. Depending on the axial force zone is determined for Pt = 0.0.
2. If the column is in "zone A", design is performed by increasing Pt and checking allowable load for that known Pt and known actual eccentricity of the column.
3. If the column is in "zone B" or in "zone C", xn is calculated for given P and Pt and checking is done for allowable moment, if allowable moment is less than the actual moment, program increases Pt and this procedure continues until the column design conditions are satisfied or the column fails as the required Pt is higher than Pt maximum value.
4. If the column is in tension, design is done by considering allowable tensile stress of steel only.
5. If biaxial design is requested program solve the following interaction equation
6. where, a = 1.0+1.66666666 * (ratio-0.2), ratio = P/Pcap & 1.0 £ a £ 2.0, Myscap, Mzscap & Pscap represents section capacity
7. If the interaction equation is not satisfied program increases Pt and calculates Pscap, Myscap and Mzscap and solve the interaction equation again and this process continues until the eqn. is satisfied or the column fails as Pt exceeds its maximum limit.
8. If biaxial design is not requested program assumes that interaction equation is satisfied (if uniaxial design is performed successfully).
9. If the interaction equation is satisfied program determines bar size and calculates no. of bars and details output is written.

D9.A.5.1 Example

UNIT KGS CMS
START CONCRETE DESIGN
CODE JAPAN
FYMAIN SRR295 ALL
FC 210 ALL
CLEAR 2.5 MEMB 2 TO 6
D9.A.6 Slab and Wall Design

To design a slab or a wall, it must first be modeled using finite elements and analyzed. The command specifications are in accordance with Chapter 2 and Chapter 6 of the Technical Reference Manual.

Elements are designed for the moments Mx and My. These moments are obtained from the element force output (see Chapter 2 of the Technical Reference Manual). The reinforcement required to resist the Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist the My moment is denoted as transverse reinforcement.

The longitudinal bar is the layer closest to the exterior face of the slab or wall. The following parameters are those applicable to slab and wall design:

1. FYMAIN — Yield stress for reinforcing steel - transverse and longitudinal.
2. FC— Concrete grade
3. CLEAR — Distance from the outer surface of the element to the edge of the bar. This is considered the same on both top and bottom surfaces of the element.
4. MINMAIN — Minimum required size of longitudinal/transverse reinforcing bar

The other parameters shown in D9.A.7 Design Parameters (on page 1973) are not applicable to slab or wall design.

D9.A.7 Design Parameters

The program contains a number of parameters which are needed to perform the design. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 10A.1 contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as centimeters and Kilograms before performing the concrete design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 173: Japanese Concrete Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as JAPAN. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2859).</td>
</tr>
<tr>
<td>BIAXIAL</td>
<td>0.0</td>
<td>Value to define biaxial or uniaxial design type for Column 0. uniaxial design only 1. design for biaxial moments</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CLEAR</td>
<td>3.0 cm (beam) 4.0 cm (Column)</td>
<td>Clear cover for Beam or clear side cover for column.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>Depth of concrete member. This value defaults to YD as provided under MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>EFACE</td>
<td>0.0</td>
<td>Face of support location at end of beam. (Note: Both SFACE &amp; EFACE are input as positive numbers).</td>
</tr>
<tr>
<td>FC</td>
<td>210 Kg/cm²</td>
<td>Compressive Strength of Concrete.</td>
</tr>
<tr>
<td>FYMAIN</td>
<td>SR235</td>
<td>Steel grade. Acceptable values for steel grade and their associated yield stress values are shown in the following table. Program automatically calculates yield stress value depending on design load type (permanent or temporary).</td>
</tr>
<tr>
<td>FYSEC</td>
<td>SR235</td>
<td>Same as FYMAIN except this is for secondary steel.</td>
</tr>
<tr>
<td>LONG</td>
<td>0.0</td>
<td>Value to define design load type 0. Permanent Loading 1. Temporary Loading</td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>41.0 cm</td>
<td>Maximum main reinforcement bar size</td>
</tr>
<tr>
<td>MAXSEC</td>
<td>41.0 cm</td>
<td>Maximum secondary reinforcement bar size.</td>
</tr>
<tr>
<td>MINMAIN</td>
<td>10 mm</td>
<td>Minimum main reinforcement bar size.</td>
</tr>
<tr>
<td>MINSEC</td>
<td>10 mm</td>
<td>Minimum secondary reinforcement bar size.</td>
</tr>
<tr>
<td>MMAG</td>
<td>1.0</td>
<td>Design moment magnification factor</td>
</tr>
<tr>
<td>NSECTION</td>
<td>12</td>
<td>Number of equally-spaced sections to be considered in finding critical moments for beam design.</td>
</tr>
</tbody>
</table>
### Design

D. Design Codes

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>REINF</td>
<td>0.0</td>
<td>Tied Column. A value of 1.0 will mean spiral.</td>
</tr>
<tr>
<td>SFACE</td>
<td>0.0</td>
<td>Face of support location at start of beam.</td>
</tr>
<tr>
<td>SMAG</td>
<td>1.0</td>
<td>Design shear magnification factor</td>
</tr>
<tr>
<td>TORSION</td>
<td>0.0</td>
<td>Value to request for torsion design for beam</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. torsion design not needed</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. torsion design needed</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Beam Design:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Critical section design results.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Five section design results &amp; design forces.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. 12 section design results &amp; design forces.</td>
</tr>
<tr>
<td>WIDTH</td>
<td>ZD</td>
<td>Beam Design:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Critical section design results.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Five section design results &amp; design forces.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. 12 section design results &amp; design forces.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Column Design:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Detail design results for critical load case only.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Design results for minimum P, maximum P, maximum MZ and maximum MY among all load cases for both ends.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Width of concrete member. This value defaults to ZD as provided under MEMBER PROPERTIES.</td>
</tr>
</tbody>
</table>

#### Table 174: Table of permissible Steel Grades and associated Yield Stresses for FYMAIN and FYSEC parameters

<table>
<thead>
<tr>
<th>Steel Grade</th>
<th>Long Term Loading</th>
<th>Short Term Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Tension &amp; Compression</td>
<td>Shear Reinforcement</td>
</tr>
<tr>
<td>SR235</td>
<td>1600</td>
<td>1600</td>
</tr>
<tr>
<td>SRR235</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SDR235</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## D9.B. Japanese Codes - Steel Design per 2005 AIJ

STAAD.Pro is capable of performing steel design based on the Japanese code AIJ 2005 *Design Standard for Steel Structures*.

### D9.B.1 General

This section presents some general statements regarding the implementation of the “Architectural Institute of Japan” (AIJ) specifications for structural steel design (2005 edition) in STAAD. The design philosophy and procedural logistics are based on the principles of elastic analysis and allowable stress design. Facilities are available for member selection as well as code checking. Two major failure modes are recognized: failure by overstressing and failure by stability considerations. The following sections describe the salient features of the design approach.

Members are proportioned to resist the design loads without exceedance of the allowable stresses or capacities and the most economical section is selected on the basis of the least weight criteria. The code checking part of the program also checks the slenderness requirements and the stability criteria. Users are recommended to adopt the following steps in performing the steel design:

- Specify the geometry and loads and perform the analysis.
- Specify the design parameter values if different from the default values.
- Specify whether to perform code checking or member selection.

The method for calculating allowable bending stress was updated for the AIJ 2005 from the AIJ 2002 code. All other allowable limit states, analysis and design methods, etc., remain unchanged. Refer to the [AIJ 2002 documentation](page 1988) for additional details.

### D9.B.2 Member Capacities

Member design and code checking per AIJ 2005 are based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. The basic measure of member capacities are

<table>
<thead>
<tr>
<th>Steel Grade</th>
<th>Long Term Loading</th>
<th>Short Term Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Tension &amp; Compression</td>
<td>Shear Reinforcement</td>
</tr>
<tr>
<td>SR295</td>
<td>1600</td>
<td>2000</td>
</tr>
<tr>
<td>SRR295</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SD295A</td>
<td>2000</td>
<td>2000</td>
</tr>
<tr>
<td>SD295B</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SDR295</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SDR345</td>
<td>2200 (2000)</td>
<td>2000</td>
</tr>
<tr>
<td>SD345</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SD390</td>
<td>2200 (2000)</td>
<td>2000</td>
</tr>
</tbody>
</table>
the allowable stresses on the member under various conditions of applied loading such as allowable tensile stress, allowable compressive stress etc. These depend on several factors such as cross sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Explained here is the procedure adopted in STAAD.Pro for calculating such capacities.

**D9.B.2.1 Design Capabilities**

All types of available shapes like H-Shape, I-Shape, L-Shapes, CHANNEL, PIPE, TUBE, etc. can be used as member property and STAAD.Pro will automatically adopt the design procedure for that particular shape if Steel Design is requested. STEEL TABLE available within STAAD.Pro or UPTABLE facility can be used for member property.

**D9.B.2.2 Methodology**

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as required by AIJ specifications. The design procedure consist of following three steps.

1. Calculation of sectional properties

   The program extracts sectional properties like sectional area (A), Moment of Inertia about Y axis and Z axis, I_{yy}, I_{zz}, from built-in steel tables and calculates the elastic moduli and radii of gyration, Z_y, Z_v, i_y, i_v using the appropriate formulas.

2. Calculation of actual and allowable stresses

   Program calculates actual and allowable stresses by following methods:

   i. Axial Stress:

      Actual tensile stresses (F_T) = \frac{\text{Force}}{\text{A x NSF}},

      NSF = \text{Net Section Factor for tension input as a design parameter}

      Actual compressive stress (F_C) = \frac{\text{Force}}{\text{A}}

      Allowable tensile stress (f_t) = \frac{\text{FYLD}}{1.5} \text{ (For Permanent Case)}

      = \frac{\text{FYLD}}{} \text{ (For Temporary Case )}

   where:

   FYLD = \text{Yield stress input as a design parameter}

   Allowable compressive stress

   \[
   f_c = \begin{cases} 
   \frac{1 - 0.4\left(\frac{\Lambda}{A}\right)^2}{\nu}F & \text{when } \lambda \leq \Lambda \\
   \frac{0.277E}{\left(\frac{\Lambda}{A}\right)^2} & \text{when } \lambda > \Lambda
   \end{cases}
   \]

   = f_c \times 1.5 \text{ (for Temporary case)}

   where:

   \[
   \Lambda = \sqrt{\frac{\pi^2E}{0.6F}}
   \]

   \[
   \nu = \frac{3}{2} + \frac{2}{3}\left(\frac{\lambda}{\Lambda}\right)^2
   \]
\( \lambda = \text{maximum slenderness, considering both principal axis} \)

\( E = \text{Young's Modulus} \)

**ii. Bending Stress:**

Actual bending stress for My for compression:

\[ (F_{bcy}) = \frac{M_y}{Z_{cy}} \]

Actual bending stress for Mz for compression

\[ (F_{bcz}) = \frac{M_z}{Z_{cz}} \]

Actual bending stress for My for tension

\[ (F_{bty}) = \frac{M_y}{Z_{ty}} \]

Actual bending stress for Mz for tension

\[ (F_{btz}) = \frac{M_z}{Z_{tz}} \]

where:

\( Z_{cy}, Z_{cz} \) are elastic section modulus about the Y and Z axis for the end in compression

\( Z_{ty}, Z_{tz} \) are elastic section modulus about the Y and Z axis for the end in tension

Allowable bending stress for My

\[ (f_{bcy}) = f_t \]

Allowable bending stress for Mz

When \( \lambda_b \leq \lambda_{pb} \), \( f_b = \frac{F}{\nu} \)

When \( \lambda_{pb} < \lambda_b \leq e_{pb} \),

\[ f_b = \frac{F}{\frac{1}{\lambda_{b}^{2}} \left( 1 - 0.4 \frac{\lambda_b - \lambda_{pb}}{e_{pb} - \lambda_{pb}} \right)} \]

When \( e_{pb} < \lambda_b \),

\[ f_b = \frac{1}{\lambda_{b}^{2}} \frac{F}{2.17} \]

where:

\[ \lambda_b = \frac{M_y}{M_e} \]

\[ e_{pb} = \frac{1}{\sqrt{0.6}} \]

\[ \nu = \frac{3}{2} + \frac{2}{3} \left( \frac{\lambda_b}{e_{pb}} \right)^2 \]

\[ M_e = C \left( \frac{n^2 EI_y E I_w}{I_y^2 b^4} + \frac{n^2 EI_y G J}{I_y^2 b^4} \right) \]

\( p_{pb} = 0.6 + 0.3 \left( \frac{M_2}{MT} \right) \) or taken as the value of if not 0

\[ C = 1.75 + 1.05 \left( \frac{M_2}{M_1} \right) + 0.3 \left( \frac{M_2}{M_1} \right)^2 \leq 2.3 \]
where

\[
\begin{align*}
M_1 &= \text{the larger of end moments about the major axis} \\
M_2 &= \text{the smaller of end moments about the major axis}
\end{align*}
\]

**Note:** \(M_2/M_1\) will be +ve for double curvature and -ve for single curvature.

For Temporary case, \(f_{bcz} = 1.5 \times (f_{bcz} \text{ for Permanent case})\)

where:

- Allowable bending stress for \(M_y, f_{bty} = f_t\)
- Allowable bending stress for \(M_z, f_{btz} = f_{bcz}\)

**Note:** The parameter \(CB\) can be used to specify a value for \(C\) directly.

**iii. Shear Stress**

Actual shear stresses are calculated by the following formula:

\[
Q_y = \frac{F_y}{A_{ww}}
\]

where:

- \(A_{ww} = \text{web shear area} = \text{depth times web thickness}\)

\[
Q_z = \frac{F_z}{A_{ff}}
\]

where:

- \(A_{ff} = \text{flange shear area} = 2/3 \times \text{total flange area}\)
- \(A_{ff} = \text{flange shear area} = 2/3 \times \text{total flange area}\)
- Allowable shear stress:
  - Permanent Loads: \(f_s = \frac{[F_y/\sqrt{3}]}{1.5}\)
  - Temporary Loads: \(f_s = \frac{F_y}{\sqrt{3}}\)
- \(F_y = \text{yield strength of steel, specified by the FYLD parameter.}\)

**3. Checking design requirements:**

User provided \(\text{RATIO}\) value (default 1.0) is used for checking design requirements:

The following conditions are checked to meet the AIJ specifications. For all the conditions calculated value should not be more than the value of \(\text{RATIO}\). If for any condition value exceeds \(\text{RATIO}\), program gives the message that the section fails.

**Conditions:**

\[
\begin{align*}
& \text{i. Axial tensile stress ratio} = \frac{F_T}{f_t} \\
& \text{ii. Axial compressive stress ratio} = \frac{F_C}{f_c} \\
& \text{iii. Combined compression & bending compressive ratio} = \frac{F_C}{f_c + F_{bcz} + F_{bcy}} \\
& \text{iv. Combined compression & bending tensile ratio} = \frac{(F_{btz} + F_{bty})}{f_t} \\
& \text{v. Combined tension & bending tensile ratio} = \frac{(F_T + F_{btz} + F_{bty})}{f_t} \\
& \text{vi. Combined tension & bending compressive ratio} = \frac{F_{bcz} + F_{bcy}}{F_{btz} + F_{bty}} \\
& \text{vii. Shear stress ratio in Y} = \frac{q_y}{f_s} \\
& \text{viii. Shear stress ratio in Z} = \frac{q_z}{f_s} \\
& \text{ix. von Mises stress ratio (if the von Mises stresses (on page 1983) were set to be checked)} = \frac{f_m}{(k \cdot f_t)}
\end{align*}
\]
Note: All other member capacities (axial tension, axial compression, and shear) are calculated as for AIJ 2002. Refer to D9.C.5 Member Capacities (on page 1993)

D9.B.3 Design Parameters

You are allowed complete control over the design process through the use of parameters in the following table. These parameters communicate design decisions from the engineer to the program. The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of the situation, some or all of these parameter values may have to be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 175: 2005 Japanese Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as JAPANESE 2005 to invoke the AIJ 2005. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>Locations of design: 0.0) design only for end moments or those at locations specified by the SECTION command. 1.0) calculate moments at twelfth points along the beam.</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>Specifies the method used for deflection checks 0) deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1) deflection check based on the principle that maximum deflection is of the cantilever type (see note a (on page 1982))</td>
</tr>
<tr>
<td>CB</td>
<td>0</td>
<td>C value from the AIJ code. Refer to D9.B.2 Member Capacities (on page 1976) - Bending Stress for how C is calculated and applied. Use 0.0 to direct the program to calculated Cb. Any other value be used in lieu of the program calculated value.</td>
</tr>
<tr>
<td>DFF</td>
<td>None(Mandatory for deflection check)</td>
<td>“Deflection Length” / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of “Deflection Length” (See note b (on page 1982))</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot; (See note b (on page 1982))</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DMAX</td>
<td>100 cm</td>
<td>Maximum allowable depth for member.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 cm</td>
<td>Minimum allowable depth for member.</td>
</tr>
<tr>
<td>FYLD</td>
<td>235 MPA</td>
<td>Yield strength of steel in Megapascal.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis. Usually, this is the minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis. Usually, this is the major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length in local y-axis to calculate slenderness ratio.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in z-axis</td>
</tr>
<tr>
<td>MAIN</td>
<td>200</td>
<td>Allowable Slenderness Limit for Compression Member</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0) suppress compression slenderness check</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Any value greater than 1 = Allowable KL/r in compression</td>
</tr>
<tr>
<td>MBG</td>
<td>0</td>
<td>Specifies how to calculate the section modulus about the Z-Z axis for H-shape, I-shape, and channel sections when performing major axis bending checks:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0) Consider the flanges and the web</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1) Consider only the flanges; the web is ignored for the calculation of the section modulus.</td>
</tr>
<tr>
<td>MISES</td>
<td>1</td>
<td>Von Mises check options:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0) Do not perform von Mises check.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1) Standard AIJ calculation. The direct stress $\sigma_x$ is determined by calculating the stress at each of the corners of the section using signed forces and the appropriate elastic modulus. The magnitude of the maximum stress is used.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2) $\tau_y$ excludes torsion stresses. The same as option 1, but in the calculation of the shear stress, the torsional moment is excluded.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3) $\sigma_x$ based on absolute forces, The direct stress $\sigma_x$ is calculated using the absolute value of the force at the section divided by the minimum of the elastic section moduli for each axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4) $\sigma_x$ based on absolute forces and $\tau_y$ excludes torsion stress. Same as option 3, but excluding the torsional moment when calculating the shear stress.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
</tbody>
</table>

For more details, refer to [D9.B.4 Von Mises Stresses Check](#) (on page 1983).
### Parameter Name | Default Value | Description
--- | --- | ---
PLB | 0 | Plastic critical slenderness ratio. If this is 0 (the default value), it will be calculated according to AIJ 2005 eqn. 5.12 or 5.14. Any other entered value will be used as the value of p-lambda-b.
RATIO | 1.0 | Permissible ratio of the actual to allowable stresses.
TMAIN | 400 | Allowable Slenderness Limit for Tension Member 1.0) suppress slenderness check Any value greater than 1 = Allowable KL/r in tension.
TRACK | 0.0 | Level of output detail: 0) Produce design summary only 1) Produce intermediate detailed output 2) Produce maximum detailed output
UNF | 1.0 | Unsupported length for calculating allowable bending stress provided as a fraction of actual member length.
UNL | Member Length | Unsupported length for calculating allowable bending stress.

### D9.B.3.1 Notes

a. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. Refer to TR.44 Printing Section Displacements for Members (on page 2846) for details on local displacement.

If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1,DZ1) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at DJ2 or the end node of the member.

Compute Delta = \( \sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2} \)

Compute Length = distance between DJ1 and DJ2 or, between start node and end node, as the case may be.

Then, if CAN is specified a value 1, \( \text{dff} = L/\Delta \)

Ratio due to deflection = \( \text{DFF}/\text{dff} \)

b. If CAN = 0, the “Deflection Length” is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The
parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.

\[ D = \text{Maximum local deflection for members 1, 2, and 3.} \]

**PARAMETERS**

- DFF 300 ALL
- DJ1 1 ALL
- DJ2 4 ALL

c. If DJ1 and DJ2 are not used, “Deflection Length” will default to the member length and local deflections will be measured from original member line.

d. The above parameters may be used in conjunction with other available parameters for steel design.

### D9.B.4 Von Mises Stresses Check

The von Mises stress equation shown below, which is modified for beam elements based on the corresponding equation in AIJ steel design code (both 2002 and 2005 editions of AIJ), indicates that the left-hand side in the equation should be less than unity. These checks are performed at locations indicated by the BEAM parameter.

**Note:** As with other design checks, the unity check value can be modified by use of the RATIO parameter.

The von Mises stresses are evaluated and checked per AIJ clause 5.16 as follows:

\[
\frac{\sqrt{\sigma_x^2 + 3\tau_{xy}^2}}{k \times f_t} < 1.0
\]

where

\[
\sigma_x = \text{Longitudinal stress in beam element. The following equation is used when the MISES parameter is set to 1 or 2. This is performed multiple times, once in each corner with the appropriate sign of the moment and value of elastic modulus. The largest stress is then used.}
\]

\[
= \frac{F_x}{A_x} + \frac{M_y}{Z_y} + \frac{M_z}{Z_z}
\]

When the MISES parameter is set to 3 or 4, then the longitudinal stress is calculated once using the smallest elastic modulus for each axis as follows.

\[
= \left| \frac{F_x}{A_x} \right| + \left| \frac{M_y}{Z_y} \right| + \left| \frac{M_z}{Z_z} \right|
\]

- \( F_x \) = axial force
- \( M_y \) = bending moment about y-axis
- \( M_z \) = bending moment about z-axis
- \( A_x \) = cross-sectional area
- \( Z_y \) = section modulus about y-axis
- \( Z_z \) = section modulus about z-axis
- \( \tau_{xy} \) = shear stress in the beam. When the MISES parameter is set to 1 or 3, this includes torsion stresses:
When the MISES parameter is set to 2 or 4, the torsion stresses are excluded:

\[
\frac{M_x}{Z_x} \pm \sqrt{\left(\frac{F_y}{A_y}\right)^2 + \left(\frac{F_z}{A_z}\right)^2}
\]

When the MISES parameter is set to 2 or 4, the torsion stresses are excluded:

\[
\frac{M_x}{Z_x} \pm \sqrt{\left(\frac{F_y}{A_y}\right)^2 + \left(\frac{F_z}{A_z}\right)^2}
\]

\[
M_x = \text{torsional moment.}
\]

\[
F_y = \text{shear stress in the y direction}
\]

\[
F_z = \text{shear stress in the z direction}
\]

\[
Z_x = \text{torsional section modulus}
\]

\[
D_x = \text{depth of the member}
\]

\[
I_x = \text{torsional constant}
\]

\[
A_y = \text{effective shear area in the y direction}
\]

\[
A_z = \text{effective shear area in the z direction}
\]

\[
f_t = \text{allowable tensile stress}
\]

\[
k = \text{loading duration factor as specified by the TMP parameter:}
\]

1.0 for permanent
1.5 for temporary

In the STRESSES output category, stress value of (numerator of the von Mises stress equation) is output as the value of \(f_m\). Along with slenderness ratios, stresses, and deflections, von Mises stress equation is checked. When its left-hand side yields the maximum ratio value, it is printed as RATIO and "VON MISES" is printed as CRITICAL COND.

D9.B.5 Verification Examples

In the next few pages are included verification examples for reference purposes.

D9.B.5.1 Verification Example No. 1

A slender, cantilever beam subjected to a load at the end. Static analysis, 3D beam element.

Problem

A cantilever beam of length 0.3 meters is subjected to a permanent joint load of 3 kN in the Y direction and 2 kN in the Z direction as well as a 0.008 kN·m torque applied at the end. Axial tension of 10 kN is also applied to the member. An H100x50x5 section is used from the Japanese steel tables.

Given

Section properties

\[
D = 100 \text{ mm}, \quad B = 50 \text{ mm}, \quad t_f = 7 \text{ mm}, \quad t_w = 5 \text{ mm}
\]

\[
I_x = 15,000 \text{ mm}^4
\]

\[
A_x = 1,185 \text{ mm}^2, \quad A_y = 500 \text{ mm}^2, \quad A_z = 467 \text{ mm}^2
\]

\[
Z_x = I_x/t_{max} = 15,000/7 = 2,143 \text{ mm}^3, \quad Z_y = 5,920 \text{ mm}^3, \quad Z_z = 37,400 \text{ mm}^3
\]

The maximum of the left hand side of the von Mises stress equation apparently occurs at the fixed end of the beam. Section forces at the fixed end are as follows:

-10.0 kN (Tension)
0.6 kN·m (Bending-Y)
0.9 kN·m (Bending-Z)
-3.0 kN (Shear-Y)
-2.0 kN (Shear-Z)
-0.008 kN·m (Torsion)

Material
FYLD = 300 MPa
E = 2.05E+05 MPa
G = E/2.6 MPa

Solution
From these section forces, \(\sigma_x\) and \(\tau_{xy}\) at the section of the fixed end are calculated as follows:

\[
\sigma_x = \frac{F_x}{A_x} + \frac{M_y}{Z_y} + \frac{M_z}{Z_z}
\]

\[
= \frac{10,000}{1,185} + \frac{600,000}{3,920} + \frac{900,000}{37,400} = 8.44 + 101.35 + 24.06 = 133.85 \text{ N/mm}^2
\]

\[
\tau_{xy} = \frac{M_x}{Z_x} + \sqrt{\left(\frac{F_y}{A_y}\right)^2 + \left(\frac{F_z}{A_z}\right)^2}
\]

\[
= \frac{-8,000}{2143} + \sqrt{\left(\frac{3,000}{500}\right)^2 + \left(\frac{2,000}{37,400}\right)^2} = 3.73 + \sqrt{6.2^2 + 4.28^2} = 11.10 \text{ N/mm}^2
\]

From \(\sigma_x\) and \(\tau_{xy}\), \(f_m\) is calculated:

\[
f_m = \sqrt{\sigma_x^2 + 3\tau_{xy}^2} = \sqrt{(133.85)^2 + 3(11.10)^2} = 135.22 \text{ N/mm}^2
\]

Since \(f_t = FYLD/1.5 = 300.0 \text{ MPa}/15 = 200.0 \text{ N/mm}^2\) and \(k = 1\) for permanent loading,

Ratio = \(135.22/(200.0 \cdot 1) = 0.676 < 1\), So OK.

Comparison

Table 176: Comparison of results for a AIJ 2005 verification example

<table>
<thead>
<tr>
<th></th>
<th>Hand Calculation</th>
<th>STAAD.Pro Result</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>von Mises Stress, (f_m) (N/mm(^2))</td>
<td>135.22</td>
<td>135.21</td>
<td>Negligible</td>
</tr>
</tbody>
</table>

STAAD Input File

STAAD SPACE VERIFICATION EXAMPLE NO.1
START JOB INFORMATION
ENGINEER DATE 18-AUG-10
END JOB INFORMATION
* VERIFICATION FOR VON MISES STRESSES IN AIJ 2005
UNIT MMS KN
JOINT COORDINATES
1 0 0 0; 2 300 0 0;
MEMBER INCIDENCES
1 1 2;
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
1 TABLE ST H100x50x5x7
UNIT MMS KN
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
UNIT METER KN
LOAD 1 LC1
JOINT LOAD
2 FX 10 FY 3 FZ 2 MX 0.008
PERFORM ANALYSIS
LOAD LIST 1
PRINT MEMBER FORCES LIST ALL
PARAMETER 1
CODE JAPANESE 2005
UNL 0.002 ALL
MISES 1 ALL
TRACK 2 ALL
FYLD 300000 ALL
CHECK CODE ALL
FINISH

Output
The TRACK 2.0 output portion is as follows:

<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Member No:       1       Profile  : ST  H100X50X5X7  (JAPANESE SECTIONS)</td>
</tr>
<tr>
<td>Ratio:    0.676  (PASS)  Reference: Eq.5.24           Loadcase:       1</td>
</tr>
<tr>
<td>----------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>

<p>| Location:    0.000  Criteria:     Stress      Load Case:     1 |
| PX:(T)     -10.000        VY:        -3.000          VZ:      -2.000 |</p>
<table>
<thead>
<tr>
<th>TX:         -0.008        MY:         0.600          MZ:      -0.900</th>
</tr>
</thead>
</table>

SECTION PROPERTIES AT DISTANCE:    0.000 MM  (UNIT: MM)
<table>
<thead>
<tr>
<th>Design D. Design Codes</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Ax: 1.18500E+03</th>
<th>Az: 4.66667E+02</th>
<th>Ay: 5.00000E+02</th>
</tr>
</thead>
<tbody>
<tr>
<td>Iz: 1.87000E+06</td>
<td>Iy: 1.48000E+05</td>
<td>J: 1.53000E+04</td>
</tr>
<tr>
<td>Zz: 3.74000E+04</td>
<td>Zy: 5.92000E+03</td>
<td>Zx: 2.18571E+03</td>
</tr>
<tr>
<td>iZ: 3.97248E+01</td>
<td>iY: 1.11756E+01</td>
<td>Cw: 3.15328E+08</td>
</tr>
</tbody>
</table>

MATERIAL PROPERTIES  (Unit: N/mm²)

| Fy: 300.000 | E: 205000.000 | G: 79000.000 |

DESIGN PROPERTIES

| Member Length: 0.300 | Lz: 0.300 | Ly: 0.300 | UNL: 0.002 |

DESIGN PARAMETERS

| Kz: 1.000 | Ky: 1.000 | NSF: 1.000 | Cb: 0.000 | plb: 0.000 |

CRITICAL SLENDERNESS (Tension)

| Actual: 26.844 | Allowable: 400.000 | Ratio: 0.067 |

CHECKS

<table>
<thead>
<tr>
<th>Tension</th>
<th>0.000</th>
<th>-10.00</th>
<th>1</th>
<th>P</th>
<th>8.44</th>
<th>200.00</th>
<th>0.042</th>
<th>Eq.5.1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>189.47</td>
<td>-</td>
<td>Eq.5.3</td>
</tr>
<tr>
<td>Bending Z (T)</td>
<td>0.000</td>
<td>-0.90</td>
<td>1</td>
<td>P</td>
<td>24.06</td>
<td>200.00</td>
<td>0.120</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Bending Z (C)</td>
<td>0.000</td>
<td>-0.90</td>
<td>1</td>
<td>P</td>
<td>24.06</td>
<td>200.00</td>
<td>0.120</td>
<td>Eq.5.7</td>
</tr>
<tr>
<td>Bending Y (T)</td>
<td>0.000</td>
<td>0.60</td>
<td>1</td>
<td>P</td>
<td>101.35</td>
<td>200.00</td>
<td>0.507</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Bending Y (C)</td>
<td>0.000</td>
<td>0.60</td>
<td>1</td>
<td>P</td>
<td>101.35</td>
<td>200.00</td>
<td>0.507</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Shear Z</td>
<td>0.000</td>
<td>-2.00</td>
<td>1</td>
<td>P</td>
<td>4.29</td>
<td>115.47</td>
<td>0.037</td>
<td>Eq.5.2</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>-3.00</td>
<td>1</td>
<td>P</td>
<td>6.00</td>
<td>115.47</td>
<td>0.052</td>
<td>Eq.5.2</td>
</tr>
<tr>
<td>Comp+ Bend C</td>
<td>0.000</td>
<td>-</td>
<td>1</td>
<td>P</td>
<td>-</td>
<td>-</td>
<td>0.627</td>
<td>Eq.6.1</td>
</tr>
</tbody>
</table>
D9.C. Japanese Codes - Steel Design per 2002 AIJ

STAAD.Pro is capable of performing steel design based on the Japanese code AIJ 2002 Design Standard for Steel Structures.

**D9.C.1 General**

The design philosophy and procedural logistics are based on the principles of elastic analysis and allowable stress design. Facilities are available for member selection as well as code checking. Two major failure modes are recognized: failure by overstressing and failure by stability considerations. The following sections describe the salient features of the design approach.

Members are proportioned to resist the design loads without exceedance of the allowable stresses or capacities and the most economical section is selected on the basis of the least weight criteria. The code checking part of the program also checks the slenderness requirements and the stability criteria. Users are recommended to adopt the following steps in performing the steel design:

- Specify the geometry and loads and perform the analysis.
- Specify the design parameter values if different from the default values.
- Specify whether to perform code checking or member selection.

**D9.C.2 Analysis Methodology**

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and in using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.
D9.C.3 Member Property Specifications

For specification of member properties of standard Japanese steel shapes, the steel section library available in STAAD may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Members properties may also be specified using the User Table facility. For more information on these facilities, refer to G.6 Member Properties (on page 2322).

D9.C.4 Built-in Japanese Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, these properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered for these members during the analysis. An example of member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained using the tools of the graphical user interface.

Following are the descriptions of different types of sections.

D9.C.4.1 I shapes

I shapes are specified in the following way:

1 250 X 125 X 10

Section-type (I)                      Web thickness (mm)
Nominal height (mm)                  Nominal width of flange (mm)

Note: While specifying the web thickness, the portion after the decimal point should be excluded.

1  TO  9  TA  ST  I300X150X11
12  TO  15  TA  ST  I350X150X9

D9.C.4.2 H shapes

H shapes are specified as follows:

H 600 X 200 X 11

Section-type (H)                      Web thickness (mm)
Nominal height (mm)                  Nominal width of flange (mm)
### Note:
While specifying the web thickness, the portion after the decimal point should be excluded.

<table>
<thead>
<tr>
<th>1 TO 8 TA ST H200X100X4</th>
</tr>
</thead>
<tbody>
<tr>
<td>13 TO 17 TA ST H350X350X12</td>
</tr>
</tbody>
</table>

#### D9.C.4.3 T shapes

T shapes are specified as follows:

```
    T 250 X 16
       Section-type (T)
               Nominal width of flange (mm)
                      Flange thickness (mm)
```

### Note:
While specifying the web thickness, the portion after the decimal point should be excluded.

<table>
<thead>
<tr>
<th>20 TO 25 TA ST T250X19</th>
</tr>
</thead>
</table>

#### D9.C.4.4 Channels

Channel sections are specified as follows:

```
    C 300 X 90 X 10
       Section-type (C)
               Nominal height (mm)
                      Nominal width of flange (mm)
                      Web thickness (mm)
```

<table>
<thead>
<tr>
<th>25 TO 34 TA ST C125X65X6</th>
</tr>
</thead>
<tbody>
<tr>
<td>46 TO 49 TA ST C200X90X8</td>
</tr>
</tbody>
</table>

#### D9.C.4.5 Double Channels

Back to back double channels, with or without a spacing in between them, are available. The letter D in front of the section name is used to specify a double channel. Front-to-front double channels are similarly added by adding FR in front of the section name.

```
    17 TO 27 TA D C300X90X10
    45 TO 76 TA D C250X90X11 SP 2.0
    28 TO 30 TA FR C200X90X8 SP 2.5
```

In the above commands, members 17 to 27 are a back-to-back double channels C300X90X10 with no spacing in between. Members 45 to 76 are a double channels C250X90X11 with a spacing of 2 length units. Members 28 to 30 are front-to-front double channels C200X90X8 with a spacing of 2.5 length units.

#### D9.C.4.6 Angles

Two types of specification may be used to describe an angle. The standard angle specification is as follows.
The letter L (signifying that the section is an angle) is followed by the length of the legs and then the thickness of the leg, all in millimeters. The word ST signifies that the section is a standard angle meaning that the major principal axis coincides with the local YY axis specified in Chapter 1 of G.4.2 Local Coordinate System (on page 2297).

```
1 4 TA ST L150X90X9
```

If the minor principal axis coincides with the local YY axis specified in Chapter 2 of the User's Manual, the word RA (Reverse Angle) should be used instead of ST as shown below.

```
7 TO 23 TA RA L90X75X9
```

**D9.C.4.7 Double angles**

Short leg back-to-back and long leg back-to-back double angles may be specified by using the words SD or LD in front of the angle size. In the case of an equal angle, either SD or LD will serve the purpose. The spacing between the angles may be specified by using the word SP after the angle size followed by the value of the spacing.

```
8 TO 25 TA SD L100X65X7 SP 2.0
36 TO 45 TA LD L300X90X11 SP 3.0
```

The first example indicates a short legs back-to-back double angle comprised of 100X65X7 angles separated by 2 length units. The latter is a long legs back-to-back double angle comprised of 300X90X11 angles separated by 3 length units.

**D9.C.4.8 Tubes**

Tube names are input by their dimensions. For example,

```
6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```

is a tube that has a height of 8 length units, width of 6 length units and a wall thickness of 0.5 length units. Only code checking, no member selection can be performed on TUBE sections.

**D9.C.4.10 Pipes (General Pipe sections)**

Circular hollow sections defined by JIS G3444:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* as general pipe sections are specified as shown in the following example.

```
1 TO 9 TA ST PIPE PIP267.4x7.0
```

specifies a pipe with outside diameter of 267.0 mm and a thickness of 7.0 mm. Only code checking, no member selection, can be performed on PIPE sections.
D9.C.4.11 Circular Hollow sections

Circular hollow sections defined by JIS G3475:2005 Design Standard for Steel Structures - Based on Allowable Stress Concept as Architectural pipe sections are specified as shown in the following example.

```
1 TO  9 TA ST PIPE CHS660.4x16
```

specifies a pipe with outside diameter of 660.4 mm and a thickness of 16.0 mm. Only code checking, no member selection, can be performed on CHS sections.

D9.C.4.12 Rectangular Hollow sections

Rectangular hollow sections defined by JIS G3466:2005 Design Standard for Steel Structures - Based on Allowable Stress Concept are specified as shown in the following example.

```
1 TO  9 TA ST PIPE RHS200x100x12
```

specifies a tube with a depth of 200 mm, a width of 100 mm, and a thickness of 12 mm. Only code checking, no member selection, can be performed on CHS sections.

D9.C.4.13 Square Hollow sections

Square hollow sections defined by JIS G3466:2005 Design Standard for Steel Structures - Based on Allowable Stress Concept are specified as shown in the following example.

```
1 TO  9 TA ST PIPE SHS200x80x12
```

specifies a square tube with a width of 200 mm and a thickness of 12 mm. Only code checking, no member selection, can be performed on CHS sections.

D9.C.4.14 Example

Sample Input file containing Japanese shapes

```
STAAD SPACE
UNIT KIP FEET
JOINT COORD
1 0 0 12 11 0 0
MEMB INCIDENT
1 1 2 11
UNIT INCH
MEMBER PROPERTY JAPANESE
* H-SHAPE
1 TA ST H200X100X4
* I SHAPE
2 TA ST I250X125X10
* T SHAPE
3 TA ST T200X19
* CHANNEL
4 TA ST C125X65X6
* DOUBLE CHANNEL
5 TA D C200X90X8
* REGULAR ANGLE
6 TA ST L100X75X7
* REVERSE ANGLE
7 TA RA L90X75X9
* DOUBLE ANGLE - LONG LEG BACK TO BACK
8 TA LD L125X75X7 SP 2.0
* DOUBLE ANGLE - SHORT LEG BACK TO BACK
9 TA SD L300X90X11 SP 1.5
```
D9.C.5 Member Capacities

Member design and code checking per AIJ 2002 are based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. The basic measure of member capacities are the allowable stresses on the member under various conditions of applied loading such as allowable tensile stress, allowable compressive stress etc. These depend on several factors such as cross sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Explained here is the procedure adopted in STAAD for calculating such capacities.

D9.C.5.1 Design Capabilities

All types of available shapes like H-Shape, I-Shape, L-Shapes, CHANNEL, PIPE, TUBE, etc. can be used as member property and STAAD will automatically adopt the design procedure for that particular shape if Steel Design is requested. STEEL TABLE available within STAAD or UPTABLE facility can be used for member property.

D9.C.5.2 Methodology

For steel design, STAAD compares the actual stresses with the allowable stresses as required by AIJ specifications. The design procedure consist of following three steps.

1. Calculation of sectional properties

The program extract sectional properties like sectional area (A), Moment of Inertia about Y axis and Z axis (Iy, Iz) from in-built Japanese Steel Table and calculates Zz, Zy, iy, iz using appropriate formula. For calculation of i (radius of gyration needed for bending), program calculates moment of inertia (Ii) and sectional area (Ai) for 1/6th section and then uses following formula:

\[ i = \sqrt{\frac{I_i}{A_i}} \]

**Note:** The above mentioned procedure for calculation of i is applicable for I shape, H shape and Channel sections.

2. Calculation of actual and allowable stresses

Allowable stresses for structural steel under permanent loading shall be determined on the basis of the values of F given in the following table.
Table 177: Table: Values of F (N/mm²)

<table>
<thead>
<tr>
<th>Thickness</th>
<th>Steel for Construction Structures</th>
<th>Steel for General Structures</th>
<th>Steel for Welded Structures</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SN400 SNR400 STKN400</td>
<td>SS400 STK400 STKR400 SSC400 SWH400</td>
<td>SS490 SS540 SM400 SMA400 SM490 Y SMA400 STKR400 STK490 SM520 SM570</td>
</tr>
<tr>
<td>t ≤ 40</td>
<td>235 325</td>
<td>235 275</td>
<td>235 325 355 400</td>
</tr>
<tr>
<td>40 &lt; t ≤ 100</td>
<td>215 295</td>
<td>215 255</td>
<td>215 295* 335 400</td>
</tr>
</tbody>
</table>

* F = 325 N/mm² when t > 75mm

Note: In checking members for temporary loading be the combination of stresses described in Chap.3, allowable stresses specified in this chapter may be increases by 50%.

Program calculates actual and allowable stresses by following methods:

i. Axial Stress:

Actual tensile stresses \( F_T \) = Force / ( A x NSF ),

NSF = Net Section Factor for tension

Actual compressive stress \( F_C \) = Force / A

Allowable tensile stress \( f_t \)

\[ f_t = \begin{cases} 
\frac{1 - 0.4(\frac{\Lambda}{\lambda})^2}{\nu} F & \text{when } \lambda \leq \Lambda \\
\frac{0.277F}{(\frac{\Lambda}{\lambda})^{3/2}} & \text{when } \lambda > \Lambda 
\end{cases} \]

\[ f_t = f_c \times 1.5 \text{ (for Temporary case)} \]

where:

\[ \Lambda = \sqrt{\frac{\pi^2 E}{0.6F}} \]
\[ v = \frac{3}{2} + \frac{2}{3}(\frac{\lambda}{A})^2 \]

ii. Bending Stress:

Actual bending stress for My for compression
\[ (F_{bcy}) = \frac{M_y}{Z_{cy}} \]
Actual bending stress for Mz for compression
\[ (F_{bcz}) = \frac{M_z}{Z_{cz}} \]
Actual bending stress for My for tension
\[ (F_{bty}) = \frac{M_y}{Z_{ty}} \]
Actual bending stress for Mz for tension
\[ (F_{btz}) = \frac{M_z}{Z_{tz}} \]

Where:

- \( Z_{cy}, Z_{cz} \) are section modulus for compression
- \( Z_{ty}, Z_{tz} \) are section modulus for tension

**Note:** The web is ignored in the calculation of Zz for H-shape, I-shape, and channel sections when the \( MGB \) parameter = 1.

Allowable bending stress for My
\[ (f_{bcy}) = f_t \]
Allowable bending stress for Mz
\[ (f_{bcz}) = \left\{ 1 - 0.4 \times \left(\frac{lb}{i}\right)^2 / (C \lambda^2) \right\} f_t \text{ max} \]
\[ = 89,000 / (lb \times h / A_f) \]
For Temporary case, \( f_{bcz} = 1.5 \times (f_{bcz} \text{ for Permanent case}) \)

Where:

\[ C = 1.75 - 1.05 \left(\frac{M_2}{M_1}\right) + 0.3 \left(\frac{M_2}{M_1}\right)^2 \]

- Allowable bending stress for M_y, \( f_{bty} = f_t \)
- Allowable bending stress for M_z, \( f_{btz} = f_{bcz} \)

**Note:** The parameter CB can be used to specify a value for C directly.

iii. Shear Stress

Actual shear stresses are calculated by the following formula:
\[ q_y = \frac{Q_y}{A_{ww}} \]
Where:

\[ A_{ww} = \text{web shear area} = \text{product of depth and web thickness} \]
\[ q_z = \frac{Q_z}{A_{ff}} \]
Where:
3. Checking design requirements:

User provided \textit{RATIO} value (default 1.0) is used for checking design requirements.

The following conditions are checked to meet the AIJ specifications. For all the conditions calculated value should not be more than the value of \textit{RATIO}. If for any condition value exceeds \textit{RATIO}, program gives the message that the section fails.

Conditions:

i. Axial tensile stress ratio = \( \frac{F_T}{f_t} \)

ii. Axial compressive stress ratio = \( \frac{F_C}{f_c} \)

iii. Combined compression & bending compressive ratio = \( \frac{F_C + F_{bcz} + F_{bcy}}{f_{bcz} + f_{bcy}} \)

iv. Combined compression & bending tensile ratio = \( \frac{F_{btz} + F_{btz}' - F_C}{f_t} \)

v. Combined tension & bending tensile ratio = \( \frac{F_{btz} + F_{btz}' - F_C}{f_t} \)

vi. Combined tension & bending compressive ratio = \( \frac{F_{bcz}}{f_{bcz}} + \frac{F_{bcy}}{f_{bcy}} \)

vii. Shear stress ratio in \( Y \) = \( \frac{q_y}{f_s} \)

viii. Shear stress ratio in \( Z \) = \( \frac{q_z}{f_s} \)

ix. von Mises stress ratio (if the \textit{von Mises stresses} (on page 2001) were set to be checked) = \( f_m/(k\cdot f_t) \)

\textit{D9.C.5.4 Allowable stress for Axial Tension}

Allowable axial stress in tension is calculated per section 5.1 (1) of the AIJ code. In members with axial tension, the tensile load must not exceed the tension capacity of the member. The tension capacity of the member is calculated on the basis of the member area. STAAD calculates the tension capacity of a given member based on a user supplied net section factor (NSF-a default value of 1.0 is present but may be altered by changing the input value, see Table 8B.1) and proceeds with member selection or code checking.

\textit{D9.C.5.5 Allowable stress for Axial Compression}

The allowable stress for members in compression is determined according to the procedure of section 5.1 (3). Compressive resistance is a function of the slenderness of the cross-section (\( K_L/r \) ratio) and the user may control the slenderness value by modifying parameters such as \( K_Y, L_Y, K_Z \) and \( L_Z \). In the absence of user provided values for effective length, the actual member length will be used. The slenderness ratios are checked against the permissible values specified in Chapter 11 of the AIJ code.

\textit{D9.C.5.6 Allowable stress for Bending}

The permissible bending compressive and tensile stresses are dependent on such factors as length of outstanding legs, thickness of flanges, unsupported length of the compression flange (UNL, defaults to member length) etc. The allowable stresses in bending (compressive and tensile) are calculated as per the criteria of Clause 5.1 (4) of the code.

\textit{D9.C.5.7 Allowable stress for Shear}

Shear capacities are a function of web depth, web thickness etc. The allowable stresses in shear are computed according to Clause 5.1 (2) of the code.
D9.C.6 Combined Loading

For members experiencing combined loading (axial force, bending and shear), applicable interaction formulas are checked at different locations of the member for all modeled loading situations. Members subjected to axial tension and bending are checked using the criteria of clause 6.2. For members with axial compression and bending, the criteria of clause 6.1 is used.

D9.C.7 Design Parameters

You are allowed complete control over the design process through the use of parameters in the following table. These parameters communicate design decisions from the engineer to the program. The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of the situation, some or all of these parameter values may have to be changed to exactly model the physical structure.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 178: 2002 Japanese Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as JAPANESE 2002 to invoke the AIJ 2002. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td>Locations of design: 0.0) design only for end moments or those at locations specified by the SECTION command. 1.0) calculate moments at twelfth points along the beam.</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>Specifies the method used for deflection checks 0) deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1) deflection check based on the principle that maximum deflection is of the cantilever type (see note a (on page 2000))</td>
</tr>
<tr>
<td>CB</td>
<td>0</td>
<td>C value from the AIJ code. Refer to D9.C.5 Member Capacities (on page 1993) Bending Stress for how C is calculated and applied. Use 0.0 to direct the program to calculated Cb. Any other value be used in lieu of the program calculated value.</td>
</tr>
<tr>
<td>DFF</td>
<td>None(Mandatory for deflection check)</td>
<td>“Deflection Length” / Maximum allowable local deflection</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of “Deflection Length” (See note b (on page 2000))</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length&quot; (See note b (on page 2000))</td>
</tr>
<tr>
<td>DMAX</td>
<td>100 cm</td>
<td>Maximum allowable depth for member.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 cm</td>
<td>Minimum allowable depth for member.</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis. Usually, this is the minor axis.</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis. Usually, this is the major axis.</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length in local y-axis to calculate slenderness ratio.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in z-axis</td>
</tr>
<tr>
<td>FYLD</td>
<td>235 MPA</td>
<td>Yield strength of steel in Megapascal.</td>
</tr>
<tr>
<td>MAIN</td>
<td>200</td>
<td>Allowable Slenderness Limit for Compression Member</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0) suppress compression slenderness check</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Any value greater than 1 = Allowable KL/r in compression</td>
</tr>
<tr>
<td>MBG</td>
<td>0</td>
<td>Specifies how to calculate the section modulus about the Z-Z axis for H-shape, I-shape, and channel sections when performing major axis bending checks:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0) Consider the flanges and the web</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1) Consider only the flanges; the web is ignored for the calculation of the section modulus.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| MISES          | 1             | Von Mises check options:  
|                |               | 0) Do not perform von Mises check.  
|                |               | 1) Standard AIJ calculation. The direct stress \(\sigma_x\) is determined by calculating the stress at each of the corners of the section using signed forces and the appropriate elastic modulus. The magnitude of the maximum stress is used.  
|                |               | 2) \(\tau_y\) excludes torsion stresses. The same as option 1, but in the calculation of the shear stress, the torsional moment is excluded.  
|                |               | 3) \(\sigma_x\) based on absolute forces, The direct stress \(\sigma_x\) is calculated using the absolute value of the force at the section divided by the minimum of the elastic section moduli for each axis.  
|                |               | 4) \(\sigma_x\) based on absolute forces and \(\tau_y\) excludes torsion stress. Same as option 3, but excluding the torsional moment when calculating the shear stress.  
|                |               | For more details, refer to D9.C.10 Von Mises Stresses Check (on page 2001). |
| NSF            | 1.0           | Net section factor for tension members. |
| RATIO          | 1.0           | Permissible ratio of the actual to allowable stresses. |
| TMAIN          | 400           | Allowable Slenderness Limit for Tension Member  
|                |               | 1.0) suppress slenderness check  
|                |               | Any value greater than 1 = Allowable KL/r in tension. |
| TRACK          | 0.0           | Level of output detail:  
|                |               | 0) Produce design summary only  
|                |               | 1) Produce intermediate detailed output  
|                |               | 2) Produce maximum detailed output |
| UNF            | 1.0           | Unsupported length for calculating allowable bending stress provided as a fraction of actual member length. |
| UNL            | Member Length | Unsupported length for calculating allowable bending stress. |
| YNG            | 0             | Method for evaluating Young's modulus, \(E\), for equation 5.8:  
|                |               | 0) Use equation 5.8 from Design Standard for Steel Structures  
|                |               | 1) Use equation SSB-1.10 of JSME |
**D9.C.7.1 Notes**

a. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. Refer to TR.44 Printing Section Displacements for Members (on page 2846) for details on local displacement.

If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let \((DX1, DY1, DZ1)\) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, \((DX2, DY2, DZ2)\) represent the deflection values at DJ2 or the end node of the member.

\[
\text{Compute } \Delta = \sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2}
\]

\[
\text{Compute Length } = \text{distance between DJ1 and DJ2 or, between start node and end node, as the case may be.}
\]

Then, if CAN is specified a value 1, \(\text{dff} = L/\Delta\)

\[
\text{Ratio due to deflection } = \frac{\text{DFF}}{\text{dff}}
\]

b. If CAN = 0, the “Deflection Length” is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.

\[D = \text{Maximum local deflection for members 1, 2, and 3.}\]

<table>
<thead>
<tr>
<th>PARAMETERS</th>
</tr>
</thead>
<tbody>
<tr>
<td>DFF 300. ALL</td>
</tr>
<tr>
<td>DJ1 1 ALL</td>
</tr>
<tr>
<td>DJ2 4 ALL</td>
</tr>
</tbody>
</table>

c. If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

d. The above parameters may be used in conjunction with other available parameters for steel design.

**D9.C.8 Code Checking**

The purpose of code checking is to check whether the provided section properties of the members are adequate to carry the forces transmitted to it by the loads on the structure. The adequacy is checked per the AIJ requirements.

Code checking is done using forces and moments at specified sections of the members. If the BEAM parameter for a member is set to 1, moments are calculated at every twelfth point along the beam, and the maximum moment about the major axis is used. When no sections are specified and the BEAM parameter is set to zero (default), design will be based on the forces at the start and end joints of the member. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from start joint) and magnitudes of the governing forces and moments are also printed.
Refer to **D1.B.1.3 Code Checking** (on page 1418) for general information on Code Checking. Refer to **TR.49 Code Checking Specification** (on page 2852) for details the specification of the Code Checking command.

**D9.C.9 Member Selection**

The member selection process basically involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments obtained from the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table.

**Note:** Member selection cannot be performed on members listed as PRISMATIC or for user-defined sections.

Refer to **D1.B.1.4 Member Selection** (on page 1419) for general information on Member Selection. Refer to **TR.49.1 Member Selection Specification** (on page 2853) for details the specification of the Member Selection command.

### Sample Input data for Steel Design

<table>
<thead>
<tr>
<th>UNIT</th>
<th>METER</th>
</tr>
</thead>
<tbody>
<tr>
<td>PARAMETER</td>
<td></td>
</tr>
<tr>
<td>CODE</td>
<td>JAPANESE 2002</td>
</tr>
<tr>
<td>NSF</td>
<td>0.85 ALL</td>
</tr>
<tr>
<td>UNL</td>
<td>10.0 MEMBER 7</td>
</tr>
<tr>
<td>KY</td>
<td>1.2 MEMBER 3 4</td>
</tr>
<tr>
<td>RATIO</td>
<td>0.9 ALL</td>
</tr>
<tr>
<td>TRACK</td>
<td>1.0 ALL</td>
</tr>
<tr>
<td>CHECK</td>
<td>CODE ALL</td>
</tr>
<tr>
<td>SELECT</td>
<td>ALL</td>
</tr>
</tbody>
</table>

**D9.C.10 Von Mises Stresses Check**

The von Mises stress equation shown below, which is modified for beam elements based on the corresponding equation in AIJ steel design code (both 2002 and 2005 editions of AIJ), indicates that the left-hand side in the equation should be less than unity. These checks are performed at locations indicated by the BEAM parameter.

**Note:** As with other design checks, the unity check value can be modified by use of the RATIO parameter.

The von Mises stresses are evaluated and checked per AIJ clause 5.16 as follows:

$$\frac{\sqrt{\sigma_x^2 + 3\tau_{xy}^2}}{k \times f_t} < 1.0$$

where

$$\sigma_x = \frac{F_x}{A_x} + \frac{M_y}{Z_y} + \frac{M_z}{Z_z}$$

Longitudinal stress in beam element. The following equation is used when the MISES parameter is set to 1 or 2. This is performed multiple times, once in each corner with the appropriate sign of the moment and value of elastic modulus. The largest stress is then used.
When the MISES parameter is set to 3 or 4, then the longitudinal stress is calculated once using the smallest elastic modulus for each axis as follows.

\[ F_{x} = \frac{F_{x}}{A_{x}} + \frac{M_{y}}{Z_{y}} + \frac{M_{z}}{Z_{z}} \]

- \( F_{x} \) = axial force
- \( M_{y} \) = bending moment about y-axis
- \( M_{z} \) = bending moment about z-axis
- \( A_{x} \) = cross-sectional area
- \( Z_{y} \) = section modulus about y-axis
- \( Z_{z} \) = section modulus about z-axis
- \( \tau_{xy} \) = shear stress in the beam. When the MISES parameter is set to 1 or 3, this includes torsion stresses:

\[ M_{x} = \frac{M_{x}}{Z_{x}} + \sqrt{\left(\frac{F_{y}}{A_{y}}\right)^2 + \left(\frac{F_{z}}{A_{z}}\right)^2} \]

When the MISES parameter is set to 2 or 4, the torsion stresses are excluded:

\[ M_{x} = \sqrt{\left(\frac{F_{y}}{A_{y}}\right)^2 + \left(\frac{F_{z}}{A_{z}}\right)^2} \]

- \( M_{x} \) = torsional moment.
- \( F_{y} \) = shear stress in the y direction
- \( F_{z} \) = shear stress in the z direction
- \( Z_{x} \) = torsional section modulus
- \( D_{x} \) = depth of the member
- \( I_{x} \) = torsional constant
- \( A_{y} \) = effective shear area in the y direction
- \( A_{z} \) = effective shear area in the z direction
- \( f_{t} \) = allowable tensile stress
- \( k \) = loading duration factor as specified by the TMP parameter:
  - 1.0 for permanent
  - 1.5 for temporary

In the STRESSES output category, stress value of (numerator of the von Mises stress equation) is output as the value of \( f_{m} \). Along with slenderness ratios, stresses, and deflections, von Mises stress equation is checked. When its left-hand side yields the maximum ratio value, it is printed as RATIO and “VON MISÉS” is printed as CRITICAL COND.

Related Links
- V.AIJ 2002 Check for MISES parameter (on page 3771)

D10. Mexican Codes

D10.A. Mexican Codes - Concrete Design per MEX NTC 1987

STAAD.Pro is capable of performing concrete design based on the Mexican code NTC 1987 Normas Técnicas Complementarias para Diseño y construcción de Estructuras de Concreto (Complementary Technical Norms for Design and Construction of Concrete Structures).
Design

D. Design Codes

Design of members per NTC 1987 requires the *STAAD Latin American Design Codes* SELECT Code Pack.

D10.A.1 Design Operations

STAAD.Pro has the capabilities for performing concrete design. It will calculate the reinforcement needed for the specified concrete section. All the concrete design calculations are based on the current: Complementary Technical Standards for the Design and Construction of Concrete Structures – Nov. 1987. (Normas Técnicas Complementarias para Diseño y construcción de Estructuras de Concreto) of the Mexican Construction Code for the Federal District – Aug. 1993 (Reglamento de Construcciones para el Distrito Federal).

D10.A.2 Section Types for Concrete Design

The following types of cross sections can be defined for concrete design.

- **Columns** — Prismatic (Rectangular, Square, and Circular)
- **Beams** — Prismatic (Rectangular & Square), Trapezoidal, and T-shapes
- **Walls** — Finite element with a specified thickness

![Concrete shape nomenclature for beams and columns](image)

*Figure 185: Concrete shape nomenclature for beams and columns*

D10.A.3 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the **MEMBER PROPERTY** command. The following example shows the required input:

```plaintext
UNIT CM
MEMBER PROPERTY
13 TO 79 PRISM YD 40. ZD 20. IZ 53333 IY 13333
11 13 PR YD 20.
14 TO 16 PRIS YD 24. ZD 48. YB 18. ZB 12.
17 TO 19 PR YD 24. ZD 18. ZB 12.
```

In the above input, the first set of members are rectangular (40 cm depth and 20 cm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 20 cm diameter. Note that no area (AX) is provided for these members. For concrete design, this property must not be provided. If shear areas and moments of inertias are not provided, the program calculates these values from YD and ZD. Notice that in the above example the IZ and IY values provided are actually 50% of the values calculated using YD and ZD. This is a conventional practice which takes into consideration revised section parameters due to cracking of section.

Note that the third and the fourth set of members in the above example represent a T-shape and a TRAPEZOIDAL shape respectively. Depending on the properties (YD, ZD, YB, ZB, etc.) provided, the program will determine whether the section is rectangular, trapezoidal or T-shaped and the BEAM design will be done accordingly.
D10.A.4 Design Parameters

The program contains a number of parameters which are needed to perform design by the Mexican code. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 3.1 is a complete list of the available parameters and their default values.

The manual describes the commands required to provide these parameters in the input file. For example, the values of SFACE and EFACE (parameters that are used in shear design), the distances of the face of supports from the end nodes of a beam, are assigned values of zero by default but may be changed depending on the actual situation. Similarly, beams and columns are designed for moments directly obtained from the analyses without any magnification. The factors MMY and MMZ may be used for magnification of column moments. For beams, the user may generate load cases which contain loads magnified by the appropriate load factors.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 179: Mexican Concrete Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as MEXICAN. Design code to follow. See [TR.53.2 Concrete Design-Parameter Specification](on page 2859).</td>
</tr>
<tr>
<td>BTP</td>
<td>2</td>
<td>Bar type to use:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. IMPERIAL (No 3 to 18)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. METRIC (4.2 to 60mm)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. MEXICAN (No 2 to 18)</td>
</tr>
<tr>
<td>CCL</td>
<td>1</td>
<td>Concrete class according to 1.4.1d) to define Modulus of Elasticity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Class 1 Concrete</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Class 2 Concrete</td>
</tr>
<tr>
<td>CFB</td>
<td>FALSE</td>
<td>Cold formed Bar classification to define development multipliers according to table 3.1 NTC</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• FALSE - Not cold formed bar</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• TRUE - Cold formed bar</td>
</tr>
<tr>
<td>CLB</td>
<td>3 cm</td>
<td>Clear cover for bottom reinforcement</td>
</tr>
<tr>
<td>CLS</td>
<td>3 cm</td>
<td>Clear cover for side reinforcement</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Parameters</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>------------</td>
</tr>
<tr>
<td>CLT</td>
<td>3 cm</td>
<td>Clear cover for top reinforcement</td>
</tr>
<tr>
<td>DAG</td>
<td>2 cm</td>
<td>Maximum diameter of aggregate, in current units.</td>
</tr>
</tbody>
</table>
| DCP            | TRUE          | Beam Loads and reactions in direct compression Cl-2.1.5.a.I 2nd paragraph  
|                |               | • FALSE - Loads applied indirectly  
|                |               | • TRUE - Direct compression |
| DEPTH          | YD            | Depth of concrete member, in current units. This value defaults to YD as provided under MEMBER PROPERTIES. |
| DIM            | TRUE          | • FALSE: Not precautions taken - Section reduction to section 1.5 NTC Concrete  
|                |               | • TRUE: Precautions are taken to assure dimensions |
| DSD            | TRUE          | Ductile frames in accordance with Section 5 of the code. Some design conditions are considered (not including, for the time being, geometric or confinement ones)  
|                |               | • FALSE - Non-Ductile frames  
|                |               | • TRUE - Ductile Frames |
| EFACE          | 0             | Face to support location of end of beam. If specified, for shear force at start is computed at a distance of EFACE+d from the start joint of the member. Positive number. |
| EXP            | FALSE         | Exposition to soil or weather to define cover and min Steel reinforcement  
|                |               | • FALSE - Not exposed to soil or weather  
<p>|                |               | • TRUE - Exposed to soil or weather |
| FC             | 200 Kg/cm²    | Compressive Strength of Concrete |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>FYMAIN</td>
<td>4,200 Kg/cm²</td>
<td>Yield Stress for main reinforcing steel</td>
</tr>
<tr>
<td>FYSEC</td>
<td>4,200 Kg/cm²</td>
<td>Yield Stress for secondary (stirrup) reinforcing steel</td>
</tr>
<tr>
<td>LSS</td>
<td>0</td>
<td>Part of the longitudinal steel considered to reduce shear. 0 (zero) is conservative. Value between 1 and 0.</td>
</tr>
<tr>
<td>LTC</td>
<td>FALSE</td>
<td>Light Concrete to define development multipliers according to table 3.1 NTC</td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>12</td>
<td>Maximum main reinforcement bar size (Number 2 -18)</td>
</tr>
<tr>
<td>MINMAIN</td>
<td>2.5</td>
<td>Minimum main reinforcement bar size (Number 2 -18)</td>
</tr>
<tr>
<td>MINSEC</td>
<td>2.5</td>
<td>Minimum secondary reinforcement bar size (Number 2 -18)</td>
</tr>
<tr>
<td>MMY</td>
<td>1.0</td>
<td>Moment magnification factor for columns, about My.</td>
</tr>
<tr>
<td>MMZ</td>
<td>1.0</td>
<td>Moment magnification factor for columns, about Mz.</td>
</tr>
<tr>
<td>MOE</td>
<td>198,000 Kg/cm²</td>
<td>Concrete modulus of elasticity.</td>
</tr>
<tr>
<td>NSECTION</td>
<td>12</td>
<td>Number of equally-spaced sections to be considered in finding critical moments for beam design</td>
</tr>
<tr>
<td>PHI</td>
<td>90 degrees</td>
<td>Stirrups angle with the axis of the element</td>
</tr>
<tr>
<td>PSS</td>
<td>TRUE</td>
<td>Slab beared perimeter. To calculate min steel required according to 2.1.2</td>
</tr>
<tr>
<td>REINF</td>
<td>0</td>
<td>Tied Column. A value of 1 will mean spiral.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Parameters</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>------------</td>
</tr>
<tr>
<td>SFACE</td>
<td>0</td>
<td>Face to support location of start of beam. If specified, for shear force at start is computed at a distance of SFACE+d from the start joint of the member. Positive number</td>
</tr>
<tr>
<td>TEQ</td>
<td>FALSE</td>
<td>Beam needed for torsional equilibrium Cl.2.1.6a) 2nd paragraph</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• FALSE - No</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• TRUE - Yes</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>Beam Design</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Critical Moment will not be printed out with beam design report.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Will mean a print out.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Will print out required steel areas for all intermediate sections specified by NSECTION.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Column Design</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Will print out detailed design results.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Will mean a print out column interaction analysis results in addition to TRACK 0 output.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. will print out a schematic interaction diagram and intermediate interaction values in addition to all of the above.</td>
</tr>
<tr>
<td>WIDTH</td>
<td>ZD</td>
<td>Width of concrete member, in current units. This value defaults to ZD as provided under MEMBER PROPERTIES</td>
</tr>
</tbody>
</table>

* These values must be provided in the current unit system being used.

**Note:** When using metric bars for design, provide values for these parameters in actual ‘mm’ units instead of the bar number. The following metric bar sizes are available: 4.2mm, 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm and 60 mm.
D10.A.5 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to locate the possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with an NSECTION parameter. All of these equally spaced sections are scanned to determine moment and shear envelopes.

D10.A.5.1 Design for Flexure

Reinforcement for positive and negative moments are calculated on the basis of the section properties provided by the user. If the section dimensions are inadequate to carry the applied load, that is if the required reinforcement is greater than the maximum allowable for the cross section, the program reports that beam fails in maximum reinforcement. Rectangular sections are also designed with compression reinforcement.

Effective depth is chosen as Total depth - (Clear cover + diameter of stirrup + half the dia. of main reinforcement), and a trial value is obtained by adopting proper bar sizes for the stirrups and main reinforcements. The relevant clauses in Sections 1.5, 1.6, 2.1.1-2-5, 3.10 and 5.2.2 of NTC Concrete are utilized to obtain the actual amount of steel required as well as the maximum allowable and minimum required steel. These values are reported as ROW, ROWMX and ROWMN in the output and can be printed using the parameter TRACK 1.0 (see D10.A.4 Design Parameters (on page 2004)). In addition, the maximum, minimum and actual bar spacing are also printed.

It is important to note that beams are designed for flexural moment MZ only. The moment MY is not considered in the flexural design.

D10.A.5.2 Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear forces are calculated at a distance (d+SFACE) and (d+EFACE) away from the end nodes of the beam. SFACE and EFACE have default values of zero unless provided under parameters (see D10.A.4 Design Parameters (on page 2004)). Note that the value of the effective depth "d" used for this purpose is the update value and accounts for the actual c.g. of the main reinforcement calculated under flexural design. Clauses 2.1.5-6 and 5.2.4 of NTC Concrete are used to calculate the reinforcement for shear forces and torsional moments. Based on the total stirrup reinforcement required, the size of bars, the spacing, the number of bars and the distance over which they are provided are calculated. Stirrups due to geometric conditions are assumed to be 2-legged, due to design conditions could be 2 or 4-legged.

D10.A.5.3 Design for Anchorage

In the output for flexural design, the anchorage details are also provided. At any particular level, the START and END coordinates of the layout of the main reinforcement is described along with the information whether anchorage in the form of a hook or continuation is required or not at these START and END points. Note that the coordinates of these START and END points are obtained after taking into account the anchorage requirements. Anchorage length is calculated on the basis of the Clauses described in Section 3.1 of NTC concrete. In case the program selects 2 different diameters for the main or compression reinforcement, only the anchorage for the largest diameter is analyzed.

D10.A.5.4 Output

<table>
<thead>
<tr>
<th>Level</th>
<th>Serial number of bar level which may contain one or more bar group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height</td>
<td>Height of bar level from the bottom of the beam</td>
</tr>
<tr>
<td>Bar Info</td>
<td>Reinforcement bar information specifying number of bars and bar size</td>
</tr>
</tbody>
</table>
From Distance from the start of the beam to the start of the reinforcement bar
To Distance from the start of the beam to the end of the reinforcement bar
Anchor (STA/END) States whether anchorage, either hook or continuation, is needed at the start (STA) or at the end (END).
Row Actually required flexural reinforcement (As/bd) where b = width of cross section (ZD for a rectangular or square section) and d = effective depth of cross section (YD minus the distance from extreme tension fiber to the centroid of main reinforcement).

ROWMN Minimum required flexural reinforcement (Amin/bd)
ROWMX Maximum required flexural reinforcement (Amax/bd)
Spacing Distance between centers of adjacent bars of main reinforcement
Vu Factored shear force at section
Vc Nominal shear strength provided by concrete
Vs Nominal shear strength provided by shear reinforcement
Tu Factored torsional moment at section
Tc Nominal torsional moment strength provided by concrete
Ts Nominal torsional moment strength provided by torsion reinforcement

D10.A.5.5 Example Output for Beam Design

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>1</td>
<td>42.</td>
<td>5 - 2.MM</td>
<td>2468.</td>
<td>6000.</td>
<td>NO</td>
<td>YES</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>212.</td>
<td>5 - 2.MM</td>
<td>0.</td>
<td>2782.</td>
<td>YES</td>
<td>NO</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

D10.A.6 Column Design

Columns design in STAAD.Pro per the Mexican code is performed for axial force and uniaxial as well as biaxial moments. All active loadings are checked to compute reinforcement. The loading which produces the largest
amount of reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and circular sections, reinforcement is always assumed to be equally distributed on all faces. This means that the total number of bars for these sections will always be a multiple of four (4). If the MMAGx & -MMAGy parameters are specified, the column moments are multiplied by the corresponding MMAG value to arrive at the ultimate moments on the column. Minimum eccentricity conditions to be satisfied according to section 2.1.3.a are checked.

Method used: Bresler Load Contour Method

Known Values: Pu, Muy, Muz, B, D, Clear cover, Fc, Fy

Ultimate Strain for concrete : 0.003

Steps involved:

1. Assume some reinforcement. Minimum reinforcement (1% for ductile design or according to section 4.2.2 ) is a good amount to start with.

2. Find an approximate arrangement of bars for the assumed reinforcement.

3. Calculate PNMAX = Po, where Po is the maximum axial load capacity of the section. Ensure that the actual nominal load on the column does not exceed PNMAX. If PNMAX is less than the axial force Pu/FR, (FR is the strength reduction factor) increase the reinforcement and repeat steps 2 and 3. If the reinforcement exceeds 6% (or 4% for ductile design), the column cannot be designed with its current dimensions.

4. For the assumed reinforcement, bar arrangement and axial load, find the uniaxial moment capacities of the column for the Y and the Z axes, independently. These values are referred to as MYCAP and MZCAP respectively.

5. Solve the Interaction Bresler equation:

\[(M_{ny}/M_{ycap})^\alpha + (M_{nz}/M_{zcap})^\alpha\]

Where \(\alpha = 1.24\). If the column is subjected to uniaxial moment: \(\alpha = 1\)

6. If the Interaction equation is satisfied, find an arrangement with available bar sizes, find the uniaxial capacities and solve the interaction equation again. If the equation is satisfied now, the reinforcement details are written to the output file.

7. If the interaction equation is not satisfied, the assumed reinforcement is increased (ensuring that it is under 6% or 4% respectively) and steps 2 to 6 are repeated.

By the moment to check shear and torsion for columns the sections have to be checked as beams and the most strict of both shear and torsion reinforcement adopted.

D10.A.7 Column Interaction

The column interaction values may be obtained by using the design parameter TRACK 1.0 or TRACK 2.0 for the column member. If a value of 2.0 is used for the TRACK parameter, 12 different Pn-Mn pairs, each representing a different point on the Pn-Mn curve are printed. Each of these points represents one of the several Pn-Mn combinations that this column is capable of carrying about the given axis, for the actual reinforcement that the column has been designed for. In the case of circular columns, the values are for any of the radial axes. The values printed for the TRACK 1.0 output are:

- P0 = Maximum allowable pure axial load on the column (moment zero).
- Pnmax = Maximum allowable axial load on the column.
- P_bal = Axial load capacity of balanced strain condition.
- M_bal = Uniaxial moment capacity of balanced strain condition.
- E_bal = M_bal / P_bal = Eccentricity of balanced strain condition.
- M0 = Moment capacity at zero axial load.
• \( P_{\text{tens}} \) = Maximum permissible tensile load on the column.
• \( \text{Des. } P_n = \frac{P_u}{FR} \) where \( FR \) is the Strength Reduction Factor and \( P_u \) is the axial load for the critical load case.
• \( \text{Des. } M_{nx} = \frac{M_{ux} \cdot M_{magx}}{FR} \) where \( FR \) is the Strength Reduction Factor and \( M_{ux} \) is the bending moment for the appropriate axis for the critical load case.
• \( M_u = \sqrt{(M_{ux} \cdot M_{magx})^2 + (M_{uy} \cdot M_{magy})^2} \)
• \( e/h = \frac{(M_n/P_n)}{h} \) where \( h \) is the length of the column

D10.A.8 Column Design Output
The next table illustrates different levels of the column design output.

<table>
<thead>
<tr>
<th>COLUMN NO.</th>
<th>1</th>
<th>DESIGN PER MEX NTC-87 - AXIAL + BENDING</th>
</tr>
</thead>
<tbody>
<tr>
<td>FY</td>
<td>411.9</td>
<td>FC - 19.6 MPa</td>
</tr>
<tr>
<td>AREA OF STEEL REQUIRED</td>
<td>=1422.857</td>
<td></td>
</tr>
<tr>
<td>BAR CONFIGURATION</td>
<td>REINF PCT.</td>
<td>LOAD</td>
</tr>
<tr>
<td>12 - NUMBER 4</td>
<td>1.693</td>
<td>1</td>
</tr>
</tbody>
</table>

(PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)

TRACK 1 generates the following additional output:

<table>
<thead>
<tr>
<th>COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KN-MET)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_0 )</td>
</tr>
<tr>
<td>1807.71</td>
</tr>
<tr>
<td>( M_0 )</td>
</tr>
<tr>
<td>66.71</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KN-MET)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_0 )</td>
</tr>
<tr>
<td>1807.71</td>
</tr>
<tr>
<td>( M_0 )</td>
</tr>
<tr>
<td>66.71</td>
</tr>
</tbody>
</table>
D10.A.9 Slab Design

Slabs are designed per Mexican NTC specifications. To design a slab, it must be modeled using finite elements.

Element design will be performed only for the moments MX and MY at the center of the element. Design will not be performed for FX, FY, FXY, MXY. Also, design is not performed at any other point on the surface of the element. Shear is checked with Q.

A typical example of element design output is shown below. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement. The parameters FYMAIN, FC, CLB, CLS, CLT, DIM, and EXP listed in D10.A.4 Design Parameters (on page 2004) are relevant to slab design. Other parameters mentioned are not used in slab design.

![Figure 186: Element moments: Longitudinal (L) and Transverse (T)](image)
**D10.A.9.1 Example Output for Element Design**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LONG. REINF (SQ.MM/MM)</th>
<th>MOM-X /LOAD (KN-MM/MM)</th>
<th>TRANS. REINF (SQ.MM/MM)</th>
<th>MOM-Y /LOAD (KN-MM/MM)</th>
</tr>
</thead>
<tbody>
<tr>
<td>47 TOP</td>
<td>0.205</td>
<td>0.00 / 0</td>
<td>0.205</td>
<td>0.00 / 0</td>
</tr>
<tr>
<td>BOTT</td>
<td>0.254</td>
<td>10.44 / 1</td>
<td>0.362</td>
<td>13.35 / 1</td>
</tr>
</tbody>
</table>

47 SHEAR CAPACITY 57.06 KN ***PASS*** FOR LOAD CASE 3

***** INDICATES REINFORCEMENT EXCEEDS MAXIMUM

***********************************END OF ELEMENT DESIGN***********************************

**D10.B. Mexican Codes - Steel Design per NTC 1987**


Design of members per NTC 1987 requires the *STAAD Latin American Design Codes SELECT Code Pack*.

**D10.B.1 General**

The design philosophy considered is that of the Load Cases and Resistance Method or Limit States Design usually known as Load and Resistance Factor Design (LRFD).

Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized—ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD.Pro implementation of the Mexican Standards for steel structures, members are proportioned to resist the design loads without exceeding the limit states of strength, and stability. It allows to check deformation to verify serviceability.

Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that main code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the Mexican specifications as implemented in STAAD.Pro steel design. A brief description of the fundamental concepts is presented here.
D10.B.2 Limit States Design Fundamentals

The primary objective of the Limit States Design Specification is to provide a uniform reliability for all steel structures under various loading conditions.

The Limit States Design Method uses separate factors for each load and resistance. Because the different factors reflect the degree of uncertainty of different loads and combinations of loads and of the accuracy of predicted strength, a more uniform reliability is possible.

The method may be summarized by the inequality

\[ Y_i Q_i \leq R_n FR \]

On the left side of the inequality, the required strength is the summation of the various load effects, \( Q_i \), multiplied by their respective load factors, \( Y_i \). The design strength, on the right side, is the nominal strength or resistance, \( R_n \), multiplied by a resistance factor, \( FR \).

In the STAAD implementation of the Mexican Standards, it is assumed that the user will use appropriate load factors and create the load combinations necessary for analysis. The design portion of the program will take into consideration the load effects (forces and moments) obtained from analysis. In calculation of resistances of various elements (beams, columns etc.), resistance (nominal strength) and applicable resistance factor will be automatically considered.

D10.B.3 Member End Forces and Moments

Member end forces and moments in the member result from loads applied to the structure. These forces are in the local member coordinate system. The following figures show the member end actions with their directions. Refer to G.18 Member End Forces (on page 2395).

D10.B.4 Section Classification

The Limit States Design specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact (type 2), noncompact (type 3), or slender element (type 4), sections depending upon their local buckling characteristics, besides sections type 1 are able for plastic design. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and design accordingly.

D10.B.5 Member in Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (D10.B.11 Design Parameters (on page 2017)), that always refers to the gross section. STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly.

In addition to the tension resistance criterion, the user defines if tension members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.
D10.B.6 Axial Compression

The column strength equations take into account inelastic deformation and other recent research in column behavior. Two equations governing column strength are available, one for inelastic buckling and the other for elastic or Euler buckling. Both equations include the effects of residual stresses and initial out-of-straightness. Compression strength for a particular member is calculated by STAAD.Pro according to the procedure outlined in Section 3.2 of the NTC. For slender elements, the procedure described in Section 2.3.6.NTC is also used.

The procedures of Section 3.2 of the Commentaries, design helps and examples of the Complementary Technical Standards for the Design and Construction of Steel Structures (de los Comentarios, ayudas de diseño y ejemplos de las Normas Técnicas Complementarias para el Diseño y Construcción de Estructuras Metálicas, DDF (Comentarios - Julio 1993) were implemented for the determination of design strength for these limit states.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ and/or LY, LZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD.Pro immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

D10.B.7 Flexural Design Strength

In the Limit States Design Method, the flexural design strength of a member is determined mainly by the limit state of lateral torsional buckling. Inelastic bending is allowed and the basic measure of flexural capacity is the plastic moment capacity of the section.

The flexural resistance is a function of plastic moment capacity, actual laterally unbraced length, limiting laterally unbraced length, buckling moment and the bending coefficient. The limiting laterally unbraced length Lu and flexural resistance Mr are functions of the section geometry and are calculated as per the procedure of Section 3.3.2 of the NTC.

The purpose of bending coefficient Cb is to account for the influence of the moment gradient on lateral-torsional buckling. This coefficient can be specified by the user through the use of parameter CB or CBy (D10.B.11 Design Parameters (on page 2017)) or may be calculated by the program (according to LRDF USA specification) if CB is specified as 0.0. In the absence of the parameter CB, a default value of 1.0 will be used.

To specify laterally unsupported length, either of the parameters UNL and UNF (see Table 10B.1) can be used.

It is taken into account the reduction of flexural resistance due to slender web according to section 4.5.8 of the NTC.

For the sections where the web and flange are slender the LRDF USA specification was used.

Stress areas due to bending about y axis (MY)
**D10.B.8 Design for Shear**

The procedure of Sect. 3.3.3 of the NTC is used in STAAD to design for shear forces in members. Besides combined bending and shear is checked according to section 3.3.4 of the NTC, considering also the limits for stiffeners of the web according to sections 4.5.6/7 of the NTC. Shear in wide flanges and channel sections is resisted by the area of the web/s.
D10.B.9 Combined Compression Axial Force and Bending

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas of the Section 3.4 of the NTC. These interaction formulas cover the general case of biaxial bending combined with axial force. They are also valid for uniaxial bending and axial force.

It is considered that the frames are part of structures that have shear walls or rigid elements so that the lateral displacements of a floor could be disregarded. The program has included formulas to include structures with lateral displacements in the future considering for B2 the columns individually and not the complete floor analysis.

It is taken into account if the elements have transverse loads and if the ends are angularly restrained.

D10.B.10 Combined Tension Axial Force and Bending

Based on Section 3.5 4 of the NTC.

D10.B.11 Design Parameters

Design per Mexican Standards is requested by using the CODE. Other applicable parameters are summarized in Table 13B.1 below. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

The parameters DMAX and DMIN may only be used for member selection only.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td></td>
<td>Must be specified as MEXICAN. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>0</td>
<td>0: Design at ends and those locations specified by SECTION command. 1: Design at ends and at every cada 1/12th point along member length</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CB</td>
<td>1</td>
<td>Coefficient C defined per section 3.3.2.2. If Cb is set to 0.0 it will be calculated by the program according to LRFD USA (CbMex=1/CbUSA). Any other value will be directly used in the design.</td>
</tr>
<tr>
<td>CMB</td>
<td>1</td>
<td>Cfactor for combined forces when there are transverse loads in the members. Section 3.4.3.3.ii of NTC</td>
</tr>
<tr>
<td>DFF</td>
<td>None(Mandatory for deflection check, TRACK 4.0)</td>
<td>&quot;Deflection Length&quot; / Maxm. allowable local deflection See Note 1 below.</td>
</tr>
<tr>
<td>DJ1</td>
<td>Start Joint of member</td>
<td>Joint No. denoting starting point for calculation of &quot;Deflection Length.&quot; See Note 1 below.</td>
</tr>
<tr>
<td>DJ2</td>
<td>End Joint of member</td>
<td>Joint No. denoting end point for calculation of &quot;Deflection Length.&quot; See Note 1 below.</td>
</tr>
<tr>
<td>DMAX</td>
<td>114 cm</td>
<td>Maximum allowable depth</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 cm</td>
<td>Minimum allowable depth</td>
</tr>
<tr>
<td>DSD</td>
<td>T</td>
<td>Perform the ductile seismic design in accordance with Section 11 (True or False). Main design conditions are considered (not including, at the moment, geometric ones)</td>
</tr>
<tr>
<td>FU</td>
<td>4,230 Kg/cm²</td>
<td>Ultimate tensile strength of steel</td>
</tr>
<tr>
<td>FYLD</td>
<td>2,530 kg/cm²</td>
<td>Minimum Yield strength of steel</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>IMM</td>
<td>0</td>
<td>Main or secondary member for the purpose of checking slenderness</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Main member</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Secondary and wind trusses</td>
</tr>
<tr>
<td>INO</td>
<td>0</td>
<td>Curve Definition according to NTC. 3.2.2.1a, defined for I shapes or tubes</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. n=1.4, laminated I shapes, tubes or built up with 3 or 4 welded plates obtained from wider plates cuts with oxygen.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. n=1, I shapes, tubes or built up with 3 or 4 welded plates</td>
</tr>
<tr>
<td>IRR</td>
<td>0</td>
<td>Variable defined for the whole structure indicating if it is regular or irregular according to section 3.4 of the NTC.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Columns that are part of regular structures</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Columns that are part of irregular structures</td>
</tr>
<tr>
<td>KX</td>
<td>1.0</td>
<td>Effective length factor for flexural-torsional buckling</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Effective length factor for local Y axis- Usually minor axis</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Effective length factor for local Z axis- Usually major axis</td>
</tr>
<tr>
<td>LDR</td>
<td>T</td>
<td>Defines if the structure has elements to bear the wind load (shear walls, wind trusses, or bracing rigid elements) that restrict lateral displacements and allow to disregard slenderness effects. (True or False)</td>
</tr>
<tr>
<td>LX</td>
<td>Member length</td>
<td>Length for determining flexural-torsional buckling</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Length to calculate slenderness ratio for buckling about local Y axis.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Length to calculate slenderness ratio for buckling about local Z axis.</td>
</tr>
<tr>
<td>NSF</td>
<td>1</td>
<td>Net section factor for tension members</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of actual load effect and design strength</td>
</tr>
<tr>
<td>STIFF</td>
<td>Longer of Member length or depth</td>
<td>Spacing of stiffeners for beams for shear design</td>
</tr>
</tbody>
</table>
| TRACK          | 0             | Controls the level of detail in output  
0. = Suppress all design strengths  
1. = Print all design strengths  
2. = Print expanded design output |
| UNB            | Member length | Unsupported length (L) of the bottom* flange for calculating flexural strength. Will be used only if compression is in the bottom flange. See Note 2 below. |
| UNT            | Member length | Unsupported length (L) of the top* flange for calculating flexural strength. Will be used only if compression is in the top flange. See Note 2 below. |

**D10.B.11.1 Notes**

1. For deflection check, parameters DFF, DJ1, and DJ2 from D3.B.6 Design Parameters (on page 1661) may be used. All requirements remain the same.
2. Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used).

**D10.B.12 Code Checking and Member Selection**

Both code checking and member selection options are available in STAAD Mexican Standards implementation.
D10.B.13 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format.

CRITICAL COND refers to the section of the Mexican NTC which governed the design.

If the TRACK is set to 1.0, member design strengths will be printed out.

D11. New Zealand Codes

D11.A. New Zealand Codes - Steel Design per NZS 3404-1997

STAAD.Pro is capable of performing steel design based on the New Zealand code NZS 3404-1997 *New Zealand Standard for Steel Structures*, Parts 1 & 2, including Amendments 1 & 2.

D11.A.1 Member Property Specifications

For specification of member properties, either the steel section library available in STAAD.Pro or the User Table facility may be used.

The following shapes are supported for design per NC3404-1997:

- I Section (rolled and welded)
- Tee cut from rolled I-section
- Single Channel
- Single Angle
- Pipe / CHS
- Tube / RHS / SHS
- Tapered I

**Note:** Using any shape not listed will result in an error message in the output and the section will not be designed.

D11.A.2 Material Properties

Any material with modulus of elasticity between 137,895 MPa (20,000 ksi) – 344738 MPa (50,000 ksi). You can design members with either built-in material constants or user-defined materials.

**Note:** If any material with modulus of elasticity value out of that range is assigned to member, it will be designed but an additional warning message will be produced.

Refer to TR.26.1 Define Material (on page 2501) for further information on the Define Material feature.
Refer to [TR.26.2 Specifying Constants for Members and Elements](on page 2503) for further information on the Built-in Material Constants feature.

Yield stress, $f_y$, and ultimate stress, $f_u$, are calculated based on steel grade table. The design parameters FYLD, FU, and SGR are used to directly specify these values.

**D11.A.3 Section Classification**

The slenderness limit for each element is calculated using table 5.2 and 6.2.4 for elements under uniform compressive stress and varying compressive stress (bending stress).

The IST design parameter related to section classification calculations.

**D11.A.4 Member Resistance**

The member resistance is calculated in STAAD according to the procedures outlined in NZS 3404-1997. Calculated design capacities are compared to corresponding axial, bending moment, and shear forces determined from the STAAD.Pro analysis. These are used to report the fail or pass status for the members designed.

**D11.A.4.1 Slenderness**

For calculating member slenderness, the length of unbraced segment for compression is considered.

For analytical member design, the MAIN and TMAIN parameters are used.

**D11.A.4.2 Bending**

Bending capacities are calculated per NZS3404 Section 5. The allowable bending moment of members is determined as the lesser of nominal section capacity and nominal member capacity (ref. Cl.5.1).

**D11.A.4.2.1 Bending Section Strength**

The nominal section moment capacity, $\phi M_s$, is calculated about both principal x and y axes and is the capacity to resist cross-section yielding or local buckling and is expressed as the product of the yield stress of the material and the effective section modulus (ref. Cl.5.2.1). The effective section modulus is a function of section type (i.e., compact, non-compact, or slender) and minimum plate yield stress $f_y$. The nominal member capacity depends on overall flexural-torsional buckling of the member (ref.Cl.5.3).

**D11.A.4.2.2 Bending Member Strength**

For sections where the web and flange yield stresses ($f_{y,web}$ and $f_{y,flange}$ respectively) are different, the lower of the two yield stresses is applied to both the web and flange to determine the slenderness of these elements.

For sections where the web and flange yield stresses ($f_{y,web}$ and $f_{y,flange}$ respectively) are different, the lower of the two yield stresses is applied to both the web and flange to determine the slenderness of these elements.

Member moment capacity, $\phi M_b$, is calculated about the principal x axis only (ref. Cl.5.6). Critical flange effective cross-section restraints and corresponding design segment and sub-segments are used as the basis for calculating capacities.

The ALM design parameter is used to specify $a_m$ (refer cl. 5.6.1.1).

The SKR, SKT, SKL, UNT, UNB, and PBRACE design parameters are also used for bending checks.
**D11.A.4.3 Shear**

Section web shear capacity, \( \phi V_v \), is calculated per Cl.5.11, including both shear yield and shear buckling capacities. Once the capacity is obtained, the ratio of the shear force acting on the cross section to the shear capacity of the section is calculated. If any of the ratios (for both local Y & Z-axes) exceed 1.0 or the allowable value provided using the RATIO parameter, the section is considered to have failed under shear.

The following table highlights which shear capacities are calculated for different profile types.

<table>
<thead>
<tr>
<th>General Profile Type</th>
<th>New Zealand Section</th>
<th>Shear Checks</th>
</tr>
</thead>
<tbody>
<tr>
<td>I-Section (i.e., parallel to minor principal axis)</td>
<td>WB, WC, UB, UC</td>
<td>Calculated for web only</td>
</tr>
<tr>
<td>T-Section</td>
<td>BT, CT</td>
<td></td>
</tr>
<tr>
<td>Channel</td>
<td>PFC</td>
<td></td>
</tr>
<tr>
<td>Angle</td>
<td>EA, UA</td>
<td>No checks performed</td>
</tr>
<tr>
<td>Tube</td>
<td>SHS, RHS</td>
<td>Calculated parallel to both z &amp; y principal axes</td>
</tr>
<tr>
<td>Pipe</td>
<td>CHS</td>
<td>Per NZS3404 5.11.4</td>
</tr>
</tbody>
</table>

Only unstiffened web capacities are calculated. Stiffened webs are not considered. Bearing capacities are not considered.

The TSP design parameter is used in shear capacity calculations.

**D11.A.4.4 Compression**

The compressive strength of members is based on limit states per NZS3404 Section 6. It is taken as the lesser of nominal section capacity and nominal member capacity.

**D11.A.4.4.1 Compression Section Strength**

Nominal section capacity, \( \phi N_s \), is a function of form factor (Cl.6.2.2), net area of the cross section, and yield stress of the material. Through the use of the NSC parameter, you may specify the net section area. Note that this parameter is different from that corresponding to tension. The program automatically calculates the form factor. The form factors are calculated based on effective plate widths per Cl.6.2.4, and the \( f_y \) yield stress is based on the minimum plate yield stress.

The NSC (net section factor for compression) design parameter is used for compression section strength checks.

**D11.A.4.4.2 Compression Bending Strength**

Nominal member capacity, \( \phi N_c \), is a function of nominal section capacity and member slenderness reduction factor (Cl.6.3.3). This value is calculated about both principal x and y axes. Here, you are required to supply the value of \( a_b \) (Cl.6.3.3) through the ALB parameter. The effective length for the calculation of compressive strength may be provided through the use of the parameters \( K_Y, K_Z, L_Y \), and \( L_Z \).

The PBCRES and LHT design parameters are also used for this check.
D11.A.4.5 Tension

The criteria governing the capacity of tension members are based on two limit states per NZS3404 Section 7.

The limit state of yielding of the gross section is intended to prevent excessive elongation of the member.

The second limit state involves fracture at the section with the minimum effective net area $\varphi N_t$ section axial tension capacities are calculated (Cl.7.2). Through the use of the NSF parameter, you may specify the net section area. STAAD.Pro calculates the tension capacity of a member based on these two limit states per Cl.7.1 and Cl.7.2 respectively of NZS3404. Eccentric end connections can be taken into account using the KT correction factor, per Cl.7.3. The $f_y$ yield stress is based on the minimum plate yield stress. Parameters FYLD, FU, and NSF are applicable for these calculations.

D11.A.4.6 Combined Forces

D11.A.4.6.1 Combined Section Strength

Combined section bending and shear capacities are calculated using the shear and bending interaction method as per Cl.5.12.3. This check is only carried out where $\varphi V_o$ section web shear capacities are calculated.

D11.A.4.6.2 Combined Member Strength

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. Here, the adequacy of a member is also examined against both section (ref. Cl.8.3.4) and member capacity (ref.Cl.8.4.5). These account for both in-plane and out-of-plane failures. If the summation of the left hand side of the equations, addressed by the above clauses, exceeds 1.0 or the allowable value provided using the RATIO parameter, the member is considered to have failed under the loading condition.

D11.A.4.7 Torsion

STAAD.Pro does not design sections or members for torsion for NZS3404.

D11.A.4.8 Seismic Provisions

The program performs the following checks per the seismic provisions in section 12 of the code.

1. Minimum specified yield stress (table 12.4)
2. Maximum ratio of $(f_y / f_u)$ (table 12.4)

    The member seismic category is specified using the DUCT design parameter.
3. Fabrication requirement (Sec.12.4.1.2) - Category 1 or 2 members shall be hot-rolled or fabricated by welding from hot-rolled plate, except that category 2 members may be cold-formed, provided that adequate ductility capacity of the member and its connections is established by experimental testing or rational design.
4. Element slenderness (sec.12.5.1.1) - The elements of category 1, 2, 3 and 4 shall comply with the plate-element slenderness limitations presented in Table 12.5.
5. Section symmetry requirement (sec.12.5.2) – The yielding regions of category 1 or 2 members shall be doubly symmetric sections. The yielding regions of category 3 members shall be doubly or singly symmetric sections.
6. Web slenderness of beam (sec.12.7.2.1) - The web thickness within the yielding region of a beam shall be not less than $(d_1/82) (f_y / 250)$ for a category 1 or 2 member or less than $(d_1/101) (f_y / 250)$ for a category 3 member.
7. Limit on axial force (sec.12.8.3.1) - The ratio of design axial force, $N^*$, to design section capacity, $\varphi N_s$, (refer to 6.2) shall not exceed the values given in (a) through (c) below:
a. The general limit given in table 12.8.1.

b. In addition to (a), for category 1, 2 and 3 column members, excluding brace members of concentrically and eccentrically braced frames, the following limitation on design axial compression shall apply, unless waived according to 12.8.3.2

\[ N^* \leq \phi N_s \left( \frac{1 + \beta_m - \sqrt{N_s / N_{o_L} - N_s / N_{o_L}}}{1 + \beta_m + \sqrt{N_s / N_{o_L} + N_s / N_{o_L}}} \right) \]  

Eq. 12.8.3.1

c. When the slenderness ratio for the member web exceeds that given in table 12.8.2 for the appropriate member category, the design axial force generated by gravity loading alone, \( N_g^* \), shall comply with the axial force limitation equation given therein.

8. Shear-bending interaction (12.10.3.1) - When a capacity design procedure is not used, at yielding regions in category 1 or 2 beams forming part of a seismic-resisting system, when designing for load combinations including earthquake loads the nominal web shear capacity of the beams shall be taken as 80% of that calculated from 5.11.4.1 and the interaction of shear and bending moment shall satisfy:

\[ M^* \leq \phi M_{sv} \]

where

\[ M_{sv} = M_s \] for \( V^* \leq 0.6\phi V_w \)

\[ M_s (1.38 - \frac{V^*}{1.6V_w}) \] for \( 0.6\phi V_w \leq V^* \leq 0.8\phi V_w \)

The ratio will be calculated as \( M^* / \phi M_{sv} \)

D11.A.5 Member Design

There are two methods available in STAAD.Pro for checking members against the requirements of NZS 3404:

- Analytical member method (referred to as MEMBER design), and
- Physical member method (referred to as PMEMBER design)

Traditionally STAAD.Pro performed code checks based on single analytical members (i.e., single members between two nodes). This implementation assumes that the design inputs (except forces and section properties) remain same throughout the length of the member. Physical member design on the other hand allows you to group single or multiple analytical members into a single physical design member for the purposes of design to NZS 3404. Thus the some of the design inputs related to member strength (unbraced lengths etc.) varies along the length of the member.

D11.A.5.1 Analytical Member Design

Thirteen equidistant cross-sections including two end sections are selected from an analytical member. For each cross-section, the member resistance checks are evaluated for each load case. The design inputs (except forces and section properties) are treated same for throughout the length of the member.

For analytical member design, by default the member is divided at 13 evenly spaced points along the member length.
D11.A.5.1.1 Automated MEMBER Design Calculations

<table>
<thead>
<tr>
<th>Automated Design Calculations</th>
<th>PMEMBER Design Parameter</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha_b$ compression member section constant per NZS3404 6.3.3.</td>
<td>n/a</td>
<td>Calculated from table 6.3.3.</td>
</tr>
<tr>
<td>$\alpha_m$ moment modification factor per NZS3404 5.6.1.1 or Table 5.6.2 depending on if the segment is restrained at on/both ends.</td>
<td>ALM</td>
<td>Calculated based on moments distribution for individual segments and sub-segments.</td>
</tr>
<tr>
<td>$f_u$ tensile strength per NZS3404 2.1.2 and AS4100 table 2.1.</td>
<td>FU</td>
<td>Based on nominal steel grade specified using SGR design parameter and element thickness.</td>
</tr>
<tr>
<td>$f_y$ yield stress per NZS3404 2.1.1 and AS4100 table 2.1.</td>
<td>FYLD</td>
<td>Based on nominal steel grade specified using SGR design parameter and element thickness.</td>
</tr>
<tr>
<td>Residual stress category for NZS3404 Table 5.2 and 6.2.4.</td>
<td>IST</td>
<td>Based on section type.</td>
</tr>
</tbody>
</table>

Related Links

- [V. NZS3404 1997-UB Section](on page 3776)

D11.A.5.2 Physical Member Design

In case of analytical member design, all the design inputs are controlled by user input which is one-time for a whole analytical member. Specially the different unbraced lengths (UNT, UNB, LZ, LY), unbraced length factors (Kz, Ky) are considered constant throughout the length of the analytical member. But in case of a physical member, these values tend not to be constant throughout the length. The real essence of physical member design in NZS3404:1997 is the consideration of different values of unbraced lengths and unbraced length factors in different locations of same physical member to replicate the actual condition. This is achieved by auto calculation of these values from the user-provided physical bracings.

In case of Physical member design in NZS3404:1997, the program does not divide the physical member into analytical member segments of equal length. Instead, each consisting analytical member is divided into 13 sections. Hence the physical member is considered as a member as a whole with “13n” sections for design checks, where n = number of component analytical members in the physical member under consideration.
Clearly, in most of the cases, the distance between two sections will not be constant throughout the length of the member. In order to prevent overlapping design sections at the ends of adjoining component analytical members, a small gap of 0.001x the member length is used from the common joint to the design section at these locations. Doing so allows the program to capture changes in internal forces, particularly shear, at these joints within the physical member.

Additionally, PMEMBER automates:

- Steel grades based on section type, and
- Tensile stress, $f_u$, and yield stress, $f_y$, values based on plate thicknesses

Therefore it is recommend that PMEMBER design be used, even when a physical member consists of only one component analytical member.

D11.A.5.2.1 Modeling with Physical Members

You can use the physical modeling tools available in the user interface Analytical Modeling and Steel Design workflows to form physical members.

You can also define physical members using the DEFINE PMEMBER command. Refer to TR.16.2 Physical Members (on page 2443) for details.

D11.A.5.2.2 Section Profiles

When a physical member is checked against the code (CODE CHECK) in NZS3404-1997, all the component analytical members within a single physical member must have the same section assigned. If not, a code check will not be made and an error is displayed in the output.

When a physical member profile is selected (SELECT) in NZS3404-1997, all the component analytical members within a single physical member must be assigned from the same profile table. For example, if one analytical member is assigned an Australian UB150X18.0, then the other analytical members must be assigned any profile from the Australian UB table.

D11.A.5.2.3 Calculation for Compression Buckling

For calculation of member compression buckling capacities about the principal axis, the PMEMBER Design uses the concept of unbraced segments against compression ($L_z$, $L_y$). The unbraced segments against compression buckling are evaluated using the user defined compression bracings (PBCRES) separately for ZZ and YY axis. The types of restraints that can be provided at any location of the PMEMBER using PBCRES are: U(unbraced), T(translational), R(rotational), and TR(translational-rotational). Compression buckling restraints automate following design calculations for each segment for buckling against both the principle axis:

- Unbraced length of a segment for compression buckling – the distance between two consecutive T/R/TR type restraint. The program calculates it only when $L_Z$ or $L_Y$ are not assigned.
- Effective length factor, $k_e$ – according to figure 4.8.3.2 of NZS3404:1997 specification using type of restraint (T/R/TR) at the ends of a segment/sub-segment. The program calculates it only when $K_Z$ and $K_Y$ are not assigned.

If you do not assign any compression buckling restraint to the physical member, the program will assume T type restraints at the member ends. But if a member is cantilever, then the program will keep the free end of the member unrestrained. Any user-provided input against member end restraints will override the values assumed by program.

Any value, other than 1.0, given to $K_Z$ and $K_Y$ parameter will replace the automatically calculated effective length factors for the entire PMEMBER.

Any value other than 0.0 (default), given to $L_Z$ or $L_Y$ parameter will replace the automatically calculated unbraced lengths for compression buckling of the entire PMEMBER.
D11.A5.2.4 Calculation for Flexural/Flange Buckling

For calculation of member bending capacities about the principal x-axis, the PMEMBER Design uses the concept of unbraced segments against flange buckling (UNT, UNB). The unbraced segments against flange buckling are evaluated using the user defined compression bracings (PBRACE) separately for TOP and BOTTOM flange. User-defined flange restraints assigned using the PBRACE design parameter are checked to see if they are effective (i.e., if they are placed on the critical flange as per NZS3404 5.5). Flange restraints not applied to the critical flange are ineffective and hence are completely ignored.

Segment layouts for PMEMBERs may change for different load cases considered for design. Some restraints may be effective for one particular load case as they are found to apply to the critical flange, however for another load case may be found not to act on the critical flange, and found to be ineffective. In other words, the critical flange can change for each load case considered.

Typically, the critical flange will be the compression flange, except for segments with a “U” restraint at one end, in which case it will be the tension flange (as is the case for a cantilever).

Design unbraced flange segments are evaluated by "F", “P”, “L”, “FR”, "PR”, or “U” effective section restraints. L restraints are considered only if are deemed to be “effective.” L restraints are only considered to be effective when positioned on the “critical” flange between F, P, FR, or FP restraints. If an L restraint is positioned on the non-critical flange it is ignored. Further, if an L restraint is positioned between a U and an F, P, FR, or FP restraint, it is ignored (regardless of whether it is on the critical or non-critical flange).

The PMEMBER Design uses the following steps to determine effective cross-section restraints for each load case considered:

i. first all user-defined restraints are checked to see if they are applied to the compression flange, with those that are not ignored

ii. next a check is made to see if a “U” type restraint is found at either end of the PMEMBER. If this is the case then any adjacent “L” restraints up to the next “F”, “FR”, “P” or “PR” restraint are also ignored, regardless of whether they are placed on the critical or non-critical flange. Refer NZS3404 5.4.2.3.

The compression flange in step i) of the routine above is calculated based on the bending moments at the locations of the restraints being considered.

<table>
<thead>
<tr>
<th>Restraint Type</th>
<th>Definition</th>
<th>Stiffness</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FR</td>
<td>Fully and rotationally restrained</td>
<td>Most stiff</td>
<td></td>
</tr>
<tr>
<td>F</td>
<td>Fully restrained</td>
<td>↓</td>
<td></td>
</tr>
<tr>
<td>PR</td>
<td>Partially and rotationally restrained</td>
<td>↓</td>
<td></td>
</tr>
<tr>
<td>P</td>
<td>Partially restrained</td>
<td>↓</td>
<td></td>
</tr>
<tr>
<td>L</td>
<td>Laterally restrained</td>
<td>↓</td>
<td>Cannot be specified at the ends of design members.</td>
</tr>
</tbody>
</table>
Segment lengths are not automatically checked to determine if they provide full lateral restraint as per NZS3404 5.3.2.4.

For design of cantilevers, the free tip should have user-defined “U” restraints applied to both top and bottom flanges.

If the effective flange restraints for any load case consist of “U” or “L” restraints only, an error will be reported.

Hence flange restraints automate following design calculations for each unbraced flange segment for bending about both the principle axis:

- Unbraced length of top flange in bending compression – only when no value is assigned to UNT
- Unbraced length of bottom flange in bending compression – only when no value is assigned to UNB
- Moment modification factor, $a_m$, per NZS3404 5.6.1.1 – only when no value is assigned to ALM
- Load height factor, $k_l$, given in Table 5.6.3(2) - only when no value is assigned to SKL
- Lateral rotation restraint factor, $k_r$, given in Table 5.6.3(3) - only when no value is assigned to SKR
- Twist restraint factor, $k_t$, given in Table 5.6.3(1) - only when no value is assigned to SKT

Notes:

a. If PMEMBER list is not provided, all the PMEMBERS are restrained by same configuration
b. It is not necessary to provide the restraint locations in sequence as the program sorts them automatically
c. Unless specified, PMEMBER ends are assumed to be Fully Restrained (F)
d. While designing any section of the member, effective restraints are searched on each side of the section along the critical flange
e. The types of restraints applied to the top and bottom flanges at each location determines the effective section restraints. These are outlined in the table below:
## Design

### D. Design Codes

<table>
<thead>
<tr>
<th>Case</th>
<th>Flange</th>
<th>Restraint on a Critical Flange</th>
<th>Restraint on a Non-Critical Flange</th>
<th>Effective Section Restraint</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>U</td>
<td>U</td>
<td>U</td>
<td></td>
</tr>
<tr>
<td>II</td>
<td>1</td>
<td>L</td>
<td>Nothing</td>
<td>L</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Nothing</td>
<td>L</td>
<td>None</td>
</tr>
<tr>
<td>III</td>
<td>1</td>
<td>P or F</td>
<td>Nothing or U</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Nothing or U</td>
<td>P or F</td>
<td>P</td>
</tr>
<tr>
<td>IV</td>
<td>1</td>
<td>PR or FR</td>
<td>Nothing or U</td>
<td>FR</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Nothing or U</td>
<td>PR or FR</td>
<td>PR</td>
</tr>
<tr>
<td>V</td>
<td>1</td>
<td>L, P or F</td>
<td>L, P, F, FR or PR</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>FR or PR</td>
<td>L, P, F, FR or PR</td>
<td>FR</td>
</tr>
</tbody>
</table>

**D11.A.5.2.5 Automated PMEMBER Design Calculations**

The NZS3404 PMEMBER Design automates many design calculations, including those required for segment/sub-segment design.

<table>
<thead>
<tr>
<th>Automated Design Calculations</th>
<th>PMEMBER Design Parameter</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>αb compression member section constant per NZS3404 6.3.3.</td>
<td>Not applicable</td>
<td>Calculated from table 6.3.3.</td>
</tr>
<tr>
<td>αm moment modification factor per NZS3404 5.6.1.1 or Table 5.6.2 depending on if the segment is restrained at on/both ends.</td>
<td>ALM</td>
<td>Calculated based on moments distribution for individual segments and sub-segments.</td>
</tr>
<tr>
<td>$f_u$ tensile strength per NZS3404 2.1.2 and AS4100 table 2.1.</td>
<td>FU</td>
<td>Based on nominal steel grade specified using SGR design parameter and element thickness.</td>
</tr>
<tr>
<td>$f_y$ yield stress per NZS3404 2.1.1 and AS4100 table 2.1.</td>
<td>FYLD</td>
<td>Based on nominal steel grade specified using SGR design parameter and element thickness.</td>
</tr>
<tr>
<td>Residual stress category for NZS3404 Table 5.2 and 6.2.4.</td>
<td>IST</td>
<td>Based on section type.</td>
</tr>
<tr>
<td>Segment and sub-segment layout for flange unbraced length.</td>
<td>PBRACE</td>
<td>Refer to the Segment or Sub-Segment for Flexural/Flange Buckling for details.</td>
</tr>
<tr>
<td>Unbraced length of top and bottom flange for flange buckling.</td>
<td>UNT, UNB</td>
<td></td>
</tr>
</tbody>
</table>
## Automated Design Calculations

<table>
<thead>
<tr>
<th>Automated Design Calculations</th>
<th>PMEMBER Design Parameter</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Segment layout for compression buckling unbraced length.</td>
<td>PBCRES</td>
<td>Refer to the Segment for Compression Buckling for details.</td>
</tr>
<tr>
<td>Effective length factor (ke) for compression buckling</td>
<td>KZ, KY</td>
<td></td>
</tr>
<tr>
<td>Unbrace length for compression buckling</td>
<td>LZ, LY</td>
<td></td>
</tr>
<tr>
<td>$k_t$ twist restraint factor as per NZS3404 Table 5.6.3(1).</td>
<td>SKT</td>
<td>Based on effective end restraints for each segment / sub-segment.</td>
</tr>
<tr>
<td>$k_l$ load height factor as per NZS3404 Table 5.6.3(2).</td>
<td>SKL, LHT</td>
<td>Based on effective end restraints for each segment / sub-segment, and LHT design parameter.</td>
</tr>
<tr>
<td>$k_r$ lateral rotation restraint factor as per NZS3404 Table 5.6.3(3).</td>
<td>SKR</td>
<td>Based on effective end restraints for each segment / sub-segment. This is where the distinction between “F” and “FR”, as well as “P” and “PR” is used.</td>
</tr>
</tbody>
</table>

### D11.A.5.2.6 Load Height Position

When LHT is set to 1.0 to specify a top flange load height position, STAAD.Pro takes the top to be the positive local y-axis of the member.

**Tip:** This may not literally be the top flange for say a column or beam with a beta angle. The local member axes can be viewed in the user interface by selecting **Beam Orientation** in the **Diagrams Labels** dialog (or press `<Ctrl+O>`).

To automate $k_l$ using NZS3404 Table 5.6.3(2), the longitudinal position of the load also needs to be considered, i.e., as either “within segment” or “at segment end.”

To determine which of these applies, the shear forces at the ends of each design segment / sub-segment is considered. If the shear force is found to have the same direction and magnitude at both ends, it is assumed that loads act at the segment end.

If on the other hand, the shear force at each end is found to have different directions or magnitudes, loads are assumed to act within the segment.

**Tip:** The above method includes an allowance for the self-weight of the member to be considered, as the self-weight always acts through the shear center.

The net sum of the end shears is also used to determine if the load is acting in the positive or negative local member y-axis direction. If LHT is set to 1.0 for top flange loading, the net sum is used to determine whether the top flange loading is acting to stabilize or destabilize the member for lateral torsional buckling. Negative local y-axis net loads act to destabilize the segments / sub-segments, whereas positive local y-axis net loads act to stabilize segments / sub-segments.
D11.A.6 Design Parameters

The design parameters outlined in the following table are used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs. The design scope indicates whether design parameters are applicable for MEMBER Design, PMEMBER Design, or both.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Table 181: New Zealand Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Design Scope</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td></td>
<td>Must be specified as NZS3404 1997 to invoke design per NZS 3404-1997. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>ALM</td>
<td>1.0</td>
<td></td>
<td>Moment modification factor (refer cl. 5.6.1.1) If ALM is 0.0 or not specified, it is automatically calculated based cl.5.6.1.1; otherwise the input value is used.</td>
</tr>
<tr>
<td>BEAM</td>
<td>1.0</td>
<td></td>
<td>Design locations: 0 = design only for end moments and those at locations specified by SECTION command. 1 = Perform design for moments at twelfth points along the beam.</td>
</tr>
<tr>
<td>DFF</td>
<td>None</td>
<td>Analytical members only</td>
<td>&quot;Deflection Length&quot;/Maximum Allowable local deflection. See DFF, DJ1, and DJ2 Parameters (on page 2037).</td>
</tr>
</tbody>
</table>
### Parameter Name | Default Value | Design Scope | Description
--- | --- | --- | ---
DJ1 | Start Joint of member | Analytical members only | Joint No. denoting start point for calculation of "deflection length"

DJ2 | End Joint of member | Analytical members only | Joint No. denoting end point for calculation of "deflection length"

DMAX | 1000.0 mm |  | Maximum allowable depth (Applicable for member selection)

DMIN | 0.0 mm |  | Minimum required depth (Applicable for member selection)

DUCT | 0 |  | Seismic category of member ( sec. 12.2.5 ):

- 0 = Non-seismic (section 12 will not be checked)
- 1 = High ductile
- 2 = Limited ductile
- 3 = Nominal ductile
- 4 = Non-ductile

FU | 500 MPa. |  | Ultimate strength of steel. If not specified, ultimate strength is calculated from SGR as per SGR Parameter (on page 2039).

FYLD | 248.0 MPa |  | Yield strength of steel. If not specified, yield strength is calculated from SGR as per SGR Parameter (on page 2039).

GLD | blank |  | Gravity load case number. By default, no load case will be selected as gravity load case.
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Design Scope</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IST</td>
<td>Refer to IST Parameter (on page 2038).</td>
<td></td>
<td>Residual stress category: 1 - SR 2 - HR 3 - CF 4 - LW 5 - HW If not specified, IST is evaluated depending upon shape. (See IST Parameter (on page 2038) for details)</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td></td>
<td>Correction factor for distribution of forces (refer cl. 7.2)</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td></td>
<td>K value for general column flexural buckling about the local Y/Z-axis. Used to calculate slenderness ratio. If specified, overrides the value calculated by program from restraint definition (PBCRES).</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>LHT</td>
<td>0</td>
<td>Physical members only</td>
<td>Load height position as described in Table 5.6.3(2) of NZS3404:1997 0 = at Shear center 1 = At top flange (See LHT Parameter (on page 2038))</td>
</tr>
<tr>
<td>LX</td>
<td>Member Length</td>
<td></td>
<td>The distance between partial or full restraints which effectively prevent twist of the section about its centroid. (sec. 8.4.4.1.2)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Design Scope</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LY</td>
<td></td>
<td></td>
<td>Length for general column flexural buckling about the local Y/Z-axis. Used to calculate slenderness ratio. If specified, overrides the value calculated by program from restraint definition (PBCRES).</td>
</tr>
<tr>
<td>LZ</td>
<td></td>
<td></td>
<td>A value less than or equal to 1.0 suppresses the slenderness ratio check. Checks are not explicitly required per NZS3404. Any value greater than 1.0 is used as the limit for slenderness in compression.</td>
</tr>
<tr>
<td>MAIN</td>
<td>180.0</td>
<td></td>
<td>Net section factor for compression members = An / Ag (refer cl. 6.2.1)</td>
</tr>
<tr>
<td>NSC</td>
<td>1.0</td>
<td></td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td></td>
<td>Physical members only</td>
</tr>
<tr>
<td>PBRACE</td>
<td>None</td>
<td></td>
<td>Refer to PBRACE Parameter (on page 2040) for details on the PBRACE parameter.</td>
</tr>
<tr>
<td>PBCRES</td>
<td>None</td>
<td></td>
<td>Refer to PBCRES Parameter (on page 2040) for details on the PBCRES parameter.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td></td>
<td>Permissible ratio of actual load effect to the design strength.</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Design Scope</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| SGR            | 0             |              | Steel Grade.  
|                |               |              | 0 = Default (see note)  
|                |               |              | 1 = AS 3679.1 - 350  
|                |               |              | 2 = AS 3679.1 - 300  
|                |               |              | (default for standard rolled sections)  
|                |               |              | 3 = AS 3679.1 - 250  
|                |               |              | 4 = AS 1163 - C450  
|                |               |              | 5 = AS 1163 - C350  
|                |               |              | 6 = AS 1163 - C250  
|                |               |              | (default for tube sections)  
|                |               |              | 7 = AS 3678 - 450  
|                |               |              | 8 = AS 3678 - 400  
|                |               |              | 9 = AS 3678 - 350  
|                |               |              | 10 = AS 3678 - WR350  
|                |               |              | 11 = AS 3678 - 300  
|                |               |              | (default for welded profiles and UPT)  
| **Note:** Refer to SGR Parameter (on page 2039) |
| SKL            | 1.0           |              | A load height factor given in Table 5.6.3(2). If not specified or specified as 0.0, will be automatically calculated. |
| SKR            | 1.0           |              | A lateral rotation restraint factor given in Table 5.6.3(3). If not specified or specified as 0.0, will be automatically calculated. |
| SKT            | 1.0           |              | A twist restraint factor given in Table 5.6.3(1). If not specified or specified as 0.0, will be automatically calculated. |
### Parameter Name | Default Value | Design Scope | Description |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>TMAIN</strong></td>
<td>400.0</td>
<td></td>
<td>A value less than or equal to 1.0 suppresses the slenderness ratio check. Checks are not explicitly required per NZS3404. Any value greater than 1.0 is used as the limit for slenderness in tension.</td>
</tr>
</tbody>
</table>
| **TRACK**        | 0            |              | Output detail
0 = report only minimum design results
1 = report design strengths in addition to TRACK 0 output
2 = provide full details of design |
| **TSP**          | 0.0          |              | Spacing between transverse stiffener provided in the web. If any value less than or equal to 0.0 is specified, clear depth of web will be used. |
| **UNB**          | Member Length|              | Unsupported length in bending compression of the bottom and top flange respectively for calculating moment resistance. If not equal to member length, overrides the value calculated by program from restraint definition (PBRACE). |
| **UNT**          | Member Length|              | |

**DFF, DJ1, and DJ2 Parameters**

(Analytical members only) Deflection calculations are not applicable to PMEMBERs.

Compute Delta = \(\sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2}\)

Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

- A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.
For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.

PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL

D = Maximum local deflection for members 1, 2, and 3.

- If DJ1 and DJ2 are not used, “Deflection Length” will default to the member length and local deflections will be measured from original member line.
- It is important to note that unless a DFF value is specified, STAAD.Pro will not perform a deflection check. This is in accordance with the fact that there is no default value for DFF.

**IST Parameter**

IST parameter is used to specify the residual stress category referred to in table 5.2 and 6.2.4.

If IST is specified, then that value is used. But if not specified then residual stress category is automated from shape of the section according to Note 1 of table 5.2.

<table>
<thead>
<tr>
<th>Profile</th>
<th>Residual stress Category</th>
</tr>
</thead>
<tbody>
<tr>
<td>Australian UB, UC, Angle, Channel, Tee, SHS, RHS, CHS</td>
<td>HR-hot-rolled or hot-finished</td>
</tr>
<tr>
<td>Australian cold formed shapes</td>
<td>CF-cold formed</td>
</tr>
<tr>
<td>Any other Australian shape</td>
<td>HW-heavily welded longitudinally</td>
</tr>
<tr>
<td>Any shape from any country other than Australia</td>
<td></td>
</tr>
</tbody>
</table>

**LHT Parameter**

If the shear force is constant within the segment, longitudinal position of the load is assumed to be at the segment end.

If there is any variation of the shear force and the load is acting downward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU), K_I is taken to be 1.4.

If there is any variation of the shear force and the load is acting upward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU), K_I is taken to be 1.0 as the load acting at the top flange is contributing to stabilize against local torsional buckling.
### SGR Parameter

NZS3404 defines the values of steel grades that are used as either normal steel or high grade steel. The following table explains the material values used when either option is specified for a particular shape:

<table>
<thead>
<tr>
<th>SGR Value</th>
<th>Steel Grade Used</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Default</td>
<td>AS/NZS 3679.1 300 for rolled sections, AS 1163 C250 for hollow sections (Pipe, Tube, HSS), AS 3678 300 for welded sections (WB, WC, UPT, Tapered)</td>
</tr>
<tr>
<td>1</td>
<td>AS/NZS 3679.1 350</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>AS/NZS 3679.1 300</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>AS/NZS 3679.1 250</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>AS 1163 C450</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>AS 1163 C350</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>AS 1163 C250</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>AS 3678 450</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>AS 3678 400</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>AS 3678 350</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>AS 3678 WR350</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>AS 3678 300</td>
<td></td>
</tr>
</tbody>
</table>

If a value for the FYLD parameter has been specified, then that value will be used. Otherwise, the SGR value will be used to determine the yield strength and tensile strength values for the steel, based on maximum thickness of the individual elements of the section. Only for shear capacity calculation web thickness is used. Similarly, Tensile Strength is determined either from FU parameter or from SGR parameter.

A check is introduced to see if yield stress is less than 100 MPa or more than 450 MPa or not. If less than 100 MPa, a warning is issued but the yield stress will not be modified. If more than 450 MPa, then also a warning is issued and the yield stress is set to 450 MPa.

#### Example of Member Design

The following example uses the Member design facility in STAAD.Pro. However, it is strongly recommended to use the Physical member design capabilities for NZS3404:

```
PARAMETER 1
CODE NZS3404 1997
ALB 0.0 MEMBER ALL
ALM 1.13 MEMBER ALL
BEAM 1.0 MEMBER ALL
```

**STAAD.Pro**

*2039 User Manual*
**PBRACE Parameter**

PBRACE {TOP | BOTTOM} f1 r1 f2 r2 ... f52 r52 (PMEMB pmember-list)

where:

- **fn** a fraction of the PMEMBER length where restraint condition is being specified. This value is any ratio between 0.0 and 1.0.
- **rn** one of the possible restraint condition:
  
  U
  F
  P
  L
  FR
  PR
  C

**PBCRES Parameter**

PBCRES {ZZ | YY} f1 r1 f2 r2 ... f52 r52 (PMEMB pmember-list)

where:

- **fn** a fraction of the PMEMBER length where restraint condition is being specified. This value is any ratio between 0.0 and 1.0.
- **rn** one of the possible restraint condition:
Example of PMEMBER Design

PARAMETER 1
CODE NZS3404 1997
DMAX 0.4 PMEMBER 20 TO 25
DMIN 0.25 PMEMBER 20 TO 25
KX 0.75 PMEMBER 20 TO 25
KY 1.0 PMEMBER 20 TO 25
LX 4.5 PMEMBER 20 TO 25
LY 6.0 PMEMBER 20 TO 25
LHT 0.0 PMEMBER 20 TO 25
NSC 0.9 PMEMBER 20 TO 25
NSF 1.0 PMEMBER 20 TO 25
PBRACE BOTTOM 0.0 F 1.0 F PMEMBER 20 TO 25
PBRACE TOP 0.0 P 0.5 L 1.0 P PMEMBER 20 TO 25
PBCRES ZZ 0.0 TR 0.3 R 1.0 U
PBCRES YY 0.0 TR 1.0 U
SGR 0.0 PMEMBER 20 TO 25
TRACK 2.0 PMEMBER 20 TO 25
CHECK CODE PMEMBER 20 TO 25

Note: Parameters for PMEMBER design must be listed using member numbers. The ALL list option may not be used for PMEMBERs.

D11.A.7 Output Format

Results of code checking and member selection are presented in a tabular format. The term CRITICAL COND refers to the section of the NZS3404 specification which governs the design.

D11.A.7.1 TRACK 0

| * PMemberPMEMBER for physical member, MEMBER for analytical member Number: 1 |
| Member Section: ST PFC250 (AISC SECTIONS) |
| Status: FAIL Ratio: 4.381 Critical Load Case: 1 Location: 0.42 |
| Critical Condition: Cl.5.1 |
| Critical Design Forces: (Unit: KN METE) |
| FX: 4.448E+00 C FY: 0.000E+00 FZ: 43.563E+00 |
| MX: 0.000E+00 MY: 0.000E+00 MZ: 0.000E+00 |

D11.A.7.2 TRACK 1

| * PMemberPMEMBER for physical member, MEMBER for analytical member Number: 1 |
| Member Section: ST PFC250 (AISC SECTIONS) |
### Design

#### D11.A.7.3 TRACK 2

STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)

**MEMBER DESIGN OUTPUT FOR PMEMBERPMEMBER for physical member, MEMBER for analytical member 1**

**DESIGN Notes**

1. (*) next to a Load Case number signifies that a P-Delta analysis has not been performed for that particular Load Case; i.e. analysis does not include second-order effects.
2. ϕ = 0.9 for all the calculations [NZS3404 Table 3.4]
3. (#) next to Young’s modulus E indicates that its value is not 200000 MPa as per NZS3404 1.4.

**DESIGN SUMMARY**

**Designation: ST PFC250 (AISC SECTIONS)**

**Governing Load Case:** 1*

**Governing Criteria:** Cl.5.1

**Governing Ratio:** 4.381 *(FAIL)

**Governing Location:** 4.500 m from Start.

**SECTION PROPERTIES**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>250.0000 mm</td>
</tr>
<tr>
<td>bf</td>
<td>90.0000 mm</td>
</tr>
<tr>
<td>tf</td>
<td>15.0000 mm</td>
</tr>
<tr>
<td>tw</td>
<td>8.0000 mm</td>
</tr>
<tr>
<td>Ag</td>
<td>4520.0000 mm²</td>
</tr>
<tr>
<td>J</td>
<td>238.0000E+03 mm⁴</td>
</tr>
<tr>
<td>Iw</td>
<td>34.8250E+09 mm⁶</td>
</tr>
<tr>
<td>Iz</td>
<td>45.1000E+06 mm⁴</td>
</tr>
<tr>
<td>Sz</td>
<td>421.0000E+03 mm³ (plastic)</td>
</tr>
<tr>
<td>Zz</td>
<td>360.8000E+03 mm³ (elastic)</td>
</tr>
<tr>
<td>rz</td>
<td>99.8893E+00 mm</td>
</tr>
<tr>
<td>Iy</td>
<td>3.6400E+06 mm⁴</td>
</tr>
<tr>
<td>Sy</td>
<td>107.0000E+03 mm³ (plastic)</td>
</tr>
<tr>
<td>Zy</td>
<td>59.2834E+03 mm³ (elastic)</td>
</tr>
<tr>
<td>ry</td>
<td>28.3780E+00 mm</td>
</tr>
</tbody>
</table>

---

**Critical Load Case:** 1  
**Critical Condition:** Cl.5.1  
**Critical Design Forces:** (Unit: KN METE)

<table>
<thead>
<tr>
<th>Force</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>4.448E+00 KN</td>
</tr>
<tr>
<td>FY</td>
<td>0.000E+00 KN</td>
</tr>
<tr>
<td>FZ</td>
<td>43.563E+00 KN</td>
</tr>
<tr>
<td>MX</td>
<td>0.000E+00 KN</td>
</tr>
<tr>
<td>MY</td>
<td>0.000E+00 KN</td>
</tr>
<tr>
<td>MZ</td>
<td>0.000E+00 KN</td>
</tr>
</tbody>
</table>

**Design Notes:**

- ϕMsz = 113.670E+00 KNm  
- ϕMsy = 24.010E+00 KNm  
- ϕMbz = 43.939E+00 KNm  
- ϕVvy = 324.000E+00 KNm  
- ϕVvz = 437.400E+00 KNm  
- ϕNs = 1.220E+03 KN  
- ϕNcz = 601.788E+00 KN  
- ϕNcy = 74.742E+00 KN  
- ϕMrz = 113.670E+00 KNm  
- ϕMrz = 324.000E+00 KNm

---

**STAAD.Pro 2042 User Manual**
MATERIAL PROPERTIES
-------------------
Material Standard : AS/NZS 3679.1
Nominal Grade : 300
Residual Stress Category : HR (Hot-rolled)
E (#) : 204999.984 MPa [NZS3404 1.4]
G : 80000.000 MPa [NZS3404 1.4]
fy, flange : 300.000 MPa [NZS3404 Table 2.1]
fy, web : 320.000 MPa [NZS3404 Table 2.1]
fu : 440.000 MPa [NZS3404 Table 2.1]

BENDING
-------
Section Bending Capacity (about Z-axis)
Critical Load Case : 1*
Critical Ratio : 1.694
Critical Location : 4.500 m from Start.
Mz* = -192.5156E+00 KNm
Section Slenderness: Compact
Zez = 421.0000E+03 mm³
ϕMsz = 113.6700E+00 KNm [NZS3404 Cl.5.1]

Section Bending Capacity (about Y-axis)
Critical Load Case : 1*
Critical Ratio : 0.000
Critical Location : 0.000 m from Start.
My* = 0.0000E+00 KNm
Section Slenderness: Compact
Zey = 88.9251E+03 mm³
ϕMsy = 24.0098E+00 KNm [NZS3404 Cl.5.1]

Member Bending Capacity
Critical Load Case : 1*
Critical Ratio : 4.381
Critical Location : 4.500 m from Start.
Critical Flange Segment:
Location (Type): 0.00 m(F ) - 9.00 m(F )
Mz* = -192.5156E+00 KNm
kt = 1.00 [NZS3404 Table 5.6.3(1)]
k1 = 1.00 [NZS3404 Table 5.6.3(2)]
k' = 1.00 [NZS3404 Table 5.6.3(3)]
le = 9.00 m [NZS3404 5.6.3]
αm = 1.384 [NZS3404 5.6.1.1.1(b)(iii)]
Mo = 42.2526E+00 KNm [NZS3404 5.6.1.1.1(d)]
αsz = 0.279 [NZS3404 5.6.1.1.1(c)]
ϕMbz = 43.9390E+00 KNm (<= ϕMsz) [NZS3404 5.6.1.1.1(a)]
Section Shear Capacity (along Y-axis)

Critical Load Case :     1*
Critical Ratio     :   0.134
Critical Location  :   0.000 m from Start.
Vy*  =    43.5625E-03 KN
ϕVvy =   324.0000E-03 KN [NZS3404 5.11.2]

Section Shear Capacity (along Z-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
Vz*  =     0.0000E+00 KN
ϕVvz =   437.4000E-03 KN [NZS3404 5.11.2]

Section Compression Capacity

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
N*    =     0.0000E+00 KN
Ae    =     4.5200E+03 mm² [NZS3404 6.2.3 / 6.2.4]
kf    =     1.000 [AS 4100 6.2.2]
An    =     4.5200E+03 mm²
ϕNs   =     1.2204E+03 KN [NZS3404 6.2.1]

Member Compression Capacity (about Z-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
N*    =     0.0000E+00 KN
Unbraced Segment:
Location (Type):   0.00 m(U )-  9.00 m(U )
Lez   =      9.00 m
αb    =      0.50 [NZS3404 Table 6.3.3(1)/6.3.3(2)]
λn,z  =    98.699 [NZS3404 6.3.3]
λ,z   =   107.400 [NZS3404 6.3.3]
ε,z   =     0.959 [NZS3404 6.3.3]
αc,z  =     0.493 [NZS3404 6.3.3]
ϕNcz  = 0.6018E+3 KN [NZS3404 6.3.3]

Member Compression Capacity (about Y-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
N*    =     0.0000E+00 KN
Unbraced Segment:
Location (Type):   0.00 m(U )-  9.00 m(U )
Ley   =     9.00 m
λn,y  = 347.417 [NZS3404 6.3.3]
λ,y   = 350.403 [NZS3404 6.3.3]
ε,y = 0.569  [NZS3404 6.3.3]
αc,y = 0.061  [NZS3404 6.3.3]
ϕNcy = 0.7474E+2 KN  [NZS3404 6.3.3]

Section Tension Capacity

Critical Load Case :  1*
Critical Ratio :  0.000
Critical Location :  0.000 m from Start.
N* = 0.0000E+00 KN
kt = 0.00  [User defined]
An = 4.5200E+03 mm²
ϕNt = 1.2204E+03 KN  [NZS3404 7.2]

COMBINED BENDING AND AXIAL
--------------------
Section Combined Capacity (about Z-axis)

Critical Load Case :  1*
Critical Ratio :  1.694
Critical Location :  4.500 m from Start.
ϕMrz = 113.6700E+00 KNm  [NZS3404 8.3.2]

Section Combined Capacity (about Y-axis)

Critical Load Case :  1*
Critical Ratio :  0.000
Critical Location :  0.000 m from Start.
ϕMry = 24.0098E+00 KNm  [NZS3404 8.3.3]

Section Combined Capacity (Biaxial)

Critical Load Case :  1*
Critical Ratio :  1.694
Critical Location :  4.500 m from Start.
γ = 1.400  [NZS3404 8.3.4]

Member In-plane Capacity (about Z-axis)

Critical Load Case :  1*
Critical Ratio :  1.694
Critical Location :  4.500 m from Start.
ϕMiz = 113.6700E+00 KNm  [NZS3404 8.4.2]

Member In-plane Capacity (about Y-axis)

Critical Load Case :  1*
Critical Ratio :  0.000
Critical Location :  0.000 m from Start.
ϕMiy = 24.0098E+00 KNm  [NZS3404 8.4.2]

Member Out-of-plane Capacity (Tension)

Critical Load Case :  1*
Critical Ratio :  0.000
Critical Location :  0.000 m from Start.
Member Out-of-plane Capacity (Compression)

Critical Load Case : 1*
Critical Ratio : 0.000
Critical Location : 0.000 m from Start.

ϕMoz,c= 0.0000E+00 KNm [NZS3404 8.4.4.2]

Member Biaxial Capacity (Tension)

Critical Load Case : 1*
Critical Ratio : 0.000
Critical Location : 0.000 m from Start.

Member Biaxial Capacity (Compression)

Critical Load Case : 1*
Critical Ratio : 0.000
Critical Location : 0.000 m from Start.

******************************************************************************

D12. Norwegian Codes

D12.A. Norwegian Codes - Steel Design per NS 3472 / NPD

STAAD.Pro is capable of performing steel design based on the Norwegian code NS 3472 Steel structures. Design rules (3rd Edition) and NPD 1993 Veiledning om utforming, beregning og dimensjonering av stalkonstruksjoner. Sist enderet 1. (Guidance on the design, calculation and dimensioning of figures constructions. Revision 1).

Design of members per NS 3472 / NPD requires the STAAD ECC Super Code SELECT Code Pack.

D12.A.1 General Notes

This user manual presents a description of the design basis, parameters and theory applied to STAAD.Pro for performing code checks according to NS 3472 ref. [1] and NPD ref. [5]. The code checks include:

- stability check (buckling)
- lateral buckling check
- yield check (von Mises)
- stability check including local plate buckling of un-stiffened pipe walls according to NPD

The code check is available for the following cross-section types:

- wide flange profiles (HEA, HEB, IPE etc.)
- pipe (OD xx ID xx)
- tube (RHS, HUP)
- channel
- angle type (only RA)
- rectangular massive box (prismatic)
user table (wide flange, I-sections, tapered I, tube, channel and RA angle)

The code check is not available for the following cross-section types:

- Double angles
- Tapered tubes
- Prismatic sections with too few section parameters defined
- Other sections that are not in the “available” list above

Please note the following:

- NS 3472 and NPD code checking covered in this document are available through two separate STAAD.Pro Code check packages.
- This document is not a lecture in use of NS 3472 or NPD. This document explains how, and which parts of, the Norwegian steel codes that have been implemented in STAAD.Pro.
- When L-sections are used, the Code Check requires RA angle definition.
- Weld design is not included in the Norwegian code checks.
- The prismatic section defined in the code check (rectangular massive box) is not identical to the general prismatic profile defined in the STAAD.Pro analysis package.

EDR does not accept any liability for loss or damage from or in consequence for use of the program.

**D12.A.1.1 Nomenclature**

NS - refers to NS 3472 ref. [1]
NS2 - refers to NS 3472 ref. [6]
NPD - refers to NPD94 ref. [5]

**D12.A.1.2 References**

1. NS 3472 3.utg. 2001
   - Prosjektering av stållkonstruksjoner
   - Beregning og dimensjonering
3. NS 3472 1.utg. 1973
   - Prosjektering av stållkonstruksjoner
   - Beregning og dimensjonering
4. Roark &Young's 5th edition
5. NPD utg. 1994
6. NS 3472 2.utg.1984
   - Prosjektering av stållkonstruksjoner
   - Beregning og dimensjonering
D12.A.2 Basis for Code Checking

This section presents general information regarding the implementation of the Norwegian codes of practice for structural steel design. This manual describes the procedures and theory used for both NS and NPD.

In general NS is used for all cross sections and shapes listed in section 1 of this manual. An exception is the treatment and check of pipe members in framed structures. NS does not give specific details about the treatment of pipes. Section 3.4 explains how this is adopted when NS is selected for code checking.

The NPD however have a more thorough check of pipe members, and consider the effect of local buckling of the pipe wall in conjunction with the stability check. In addition, the NPD code gives joint capacity formulae for brace to chord connections for pipe members.

The design philosophy and procedural logistics are based on the principles of elastic analysis and ultimate limit state design. Two major failure modes are recognized:

- failure by overstressing
- failure by stability considerations

The following sections describe the salient features of the design approach. Members are proportioned to resist the design loads without exceeding the characteristic stresses or capacities and the most economic section is selected on the basis of the least weight criteria. It is generally assumed that the user will take care of the detailing requirements like the provision of stiffeners and check the local effects like flange buckling, web crippling, etc.

You are allowed complete control over the design process through the use of the parameters listed in Table 2.1. Default values of parameters will yield reasonable results in most circumstances. However, the user should control the design and verify results through the use of the design parameters.

D12.A.2.1 Calculation of Forces and Bending Moments

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary loading conditions and combinations provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary load combinations.

D12.A.2.2 Members with Axial Forces

For tension only members, axial tension capacity is checked for the ultimate limit stress. For compression members, axial compression capacity is checked in addition to lateral buckling and ultimate limit stress. The largest slenderness ratio ($\lambda$) shall not be greater than 250 according to NS 11.7 Stability is checked as per the procedure of NS 12.3. The buckling curves of NS fig. 3 have been incorporated into the STAAD.Pro code check. The coefficient $\alpha$ (as per NS Table 10) can be specified in both directions through the use of parameters CY and CZ. In the absence of parameters CY and/or CZ, default $a$-value will be according to NS table 11.

D12.A.2.3 Members with Axial Force and Bending Moments

For compression members with bending, interaction formulae of NS table 12.3.4.2 are checked for appropriate loading situation. All compression capacities are calculated per the procedure of NS 12.3.

The equivalent moment factor $\beta$ is calculated using the procedure of NS table 12. Two different approaches are used depending upon whether the members can sway or not. Conditions for sidesway and transverse loading can be specified through the use of parameters SSY and SSZ. For members that cannot sway, without transverse loading, coefficients b are calculated and proper dimensioning moments are used in the interaction formulae.
**D12.A.2.4 Lateral Buckling**

Lateral torsional buckling is checked as per the procedure of NS 12.3.4. The procedure for calculation of ideal buckling moment for sections with two axis of symmetry has been implemented. The coefficient can be provided by the user through the use of parameter CB. In the absence of CB, a value of 1.0 will be used. Torsional properties for cross sections (torsional constant and warping constant) are calculated using formulae from NS 3472. This results in slightly conservative estimates of torsional parameters. The program will automatically select the maximum moment in cases where $M_{vd}$ is less than $M_{zd}$.

**D12.A.2.5 Von Mises Yield Criterion**

Combined effect of axial, bending, horizontal/vertical shear and torsional shear stress is calculated at 13 sections on a member and up to 9 critical points at a section. The worst stress value is checked against yield stress divided by appropriate material factor. The von Mises calculates as:

$$\sigma_j = \sqrt{(\sigma_x + \sigma_{by} + \sigma_{bz})^2 + 3(\tau_x + \tau_y + \tau_z)^2} \leq \frac{f_y}{\gamma_m}$$

**D12.A.2.6 Material Factor and Nominal Stresses**

The design resistances are obtained by dividing the characteristic material strength by the material factor. NS 3472

The material factor default value is 1.10. Other values may be input with the MF parameter. The nominal stresses should satisfy

$$\sigma_j \leq \frac{f_y}{\gamma_m} = f_d$$

NPD

The general requirement is according to NPD 3.1.1. For stability the NPD 3.1.1 and 3.1.3 requires that the structural coefficient is considered.

$$S_d \leq f_{kd} = \frac{f_k}{\gamma_m \gamma_{mk}(S_d)}$$

Where:

- $S_d$ = reference stress or load effect resultant
- $f_k$ = characteristic capacity
- $f_{kd}$ = design capacity
- $\gamma_m$ = material coefficient
- $\gamma_{mk}$ = structural coefficient

$\gamma_m$ is default set to 1.10.

$\gamma_{mk}$ shall be equal to 1.0 for frames. For pipe members $\gamma_{mk}$ is a function of the reduced slenderness. In the STAAD.Pro implemented NPD code this is calculated automatically.

**D12.A.2.7 Code Checking According to NPD**

The following parts of Chapter 3 in the NPD guidelines have been implemented.
a. Control of nominal stresses. (NPD 3.1.2).
b. Buckling of pipe members in braced frames, including interaction with local shell buckling (NPD 3.2.2, 3.2.3).
c. Buckling of un-stiffened closed cylindrical shells, including interaction with overall column buckling (NPD 3.4.4, 3.4.6, 3.4.7 and 3.4.9).
d. Joint capacity check for gap as well as for overlap joints (NPD 3.5.2).

Check b) provides the unity check based on the beam-column buckling interaction formulae in NPD 3.2.2. The interaction between global and local buckling due to axial load and hydrostatic pressure is accounted for through computation of an axial characteristic capacity to replace the yield stress in the beam-column buckling formulae.

**Note:** Check b) handles members subjected to axial loads, bending moments and hydrostatic pressure. In other words, check b) assumes that stresses resulting from shear and torsion are of minor importance, e.g., in jacket braces.

Check c) provides the unity check based on the stability requirement for un-stiffened cylindrical shells subjected to axial compression or tension, bending, circumferential compression or tension, torsion or shear. The unity check refers to the interaction formulae in NPD 3.4.4.1. The stability requirement is given in NPD 3.4.7.

**Norwegian Codes - Steel Design per NS 3472 / NPD**

STAAD.Pro performs a stability check on aluminum alloys according to buckling curve in ECCS (European recommendation for aluminum alloy structures 1978). It is possible to select heat-treated or non heat-treated alloy from the parameter list in the STAAD.Pro input file.

For heat-treated use $CY = CZ = 0.1590$, and for non heat-treated use $CY = CZ = 0.2420$.

Tracks 1.0 and 9.0 print buckling curve H for heat-treated, and buckling curve N for non heat-treated. The yield check is the same as for steel.

**D12.A.3 Design Parameters**

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

**Table 182: Design Parameters for Norwegian Steel design code**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>none</td>
<td>Must be specified as either NS3472 for NS or NPD for NPD (NOR may also be used for both). Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
<td></td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
<td>Reference</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
<td>-----------</td>
</tr>
<tr>
<td>BEAM</td>
<td>0.0</td>
<td>Parameter BEAM 1.0 ALL tells the program to calculate von Mises at 13 sections along each member, and up to 8 points at each section. (Depending on what kind of shape is used.) <strong>Note:</strong> Must be set to 1.0</td>
<td>Sec. NS 12.2.2</td>
</tr>
<tr>
<td>BY</td>
<td>1.0</td>
<td>Buckling length coefficient, β for weak axis buckling (y-y) (NOTE: BY &gt; 0.0)</td>
<td>Fig. NS 3 Sec. NS 12.3</td>
</tr>
<tr>
<td>BZ</td>
<td>1.0</td>
<td>Buckling length coefficient, β, for strong axis buckling (z-z) (NOTE: BZ &gt; 0.0)</td>
<td>Fig. NS 3 Sec. NS 12.3</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Lateral buckling coefficient, Y. Used to calculate the ideal buckling moments, $M_{vi}$</td>
<td>Sec. NS 2 A5.5.2 Fig. NS2 A5.5.2a)-e)</td>
</tr>
<tr>
<td>CMY</td>
<td>1.0</td>
<td>Water depth in meters for hydrostatic pressure calculation for pipe members</td>
<td>Valid for the NPD code only</td>
</tr>
<tr>
<td>CMZ</td>
<td>0.49</td>
<td>$\alpha_{LT}$ for sections in connection with lateral buckling</td>
<td>Sec. NS 12.3.4 Fig. NS 6.</td>
</tr>
<tr>
<td>CY/CZ</td>
<td>Default see NS 3472</td>
<td>Buckling curve coefficient, a about local z-axis (strong axis). Represent the a, a₀, b, c, d curve.</td>
<td>Fig. NS 3 Sec. NS 12.2 NS Table 11</td>
</tr>
<tr>
<td>DMAX</td>
<td>100.0 [cm]</td>
<td>Maximum allowable depth of steel section.</td>
<td></td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0 [cm]</td>
<td>Minimum allowable depth of steel section.</td>
<td></td>
</tr>
<tr>
<td>FYLD</td>
<td>235</td>
<td>Yield strength of steel, $f_y$ (St37) [N/mm²]</td>
<td>Tab. NS 3</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
<td>Reference</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
<td>-----------</td>
</tr>
<tr>
<td>MF</td>
<td>1.1 (NS3472) 1.15 (NPD)</td>
<td>Material factor / Resistance factor, $\gamma_m$</td>
<td>Sec. NS 10.4.2 Sec. NPD 3.1</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
<td>Sec. NS 12.3.4.2</td>
</tr>
<tr>
<td>SSY</td>
<td>0.0</td>
<td>$0.0 = \text{No sidesway. } \beta = \text{calculated. } &gt; 0.0 = \text{Sidesway in local y-axis weak axis } \beta_m = \text{SSY}$</td>
<td>Sec. NS 12.3.4 Tab. NS 12 Sec. NPD 3.2.1.4</td>
</tr>
<tr>
<td>SSZ</td>
<td>0.0</td>
<td>$0.0 = \text{No sidesway. } \beta = \text{calculated. } &gt; 0.0 = \text{Sidesway in local y-axis weak axis } \beta_m$</td>
<td>Sec. NS 12.3.4 Tab. NS 12 Sec. NPD 3.2.1.4</td>
</tr>
</tbody>
</table>
| TRACK          | 0.0           | Controls the level of detail in the output:  
0.0 = Suppress critical member stresses. 
1.0 = Print all critical member stresses, i.e., DESIGN VALUES 
2.0 = Print von Mises stresses. 
9.0 = Large output, 1 page for each member. 
Refer to D12A8 Tabulated Results (on page 2082) for complete list of available TRACKs and print examples. | |
| UNL            | Member length | Effective length for lateral buckling calculations (specify buckling length). Distance between fork supports or between effective side supports for the beam | Sec. NS 12.3 |

The parameter CMY will, when given with negative value, define an inside pressure in pipe members. The pressure corresponds to given water depth in meters.

The parameter CB defines the $\varphi$ value with respect to calculation of the ideal lateral buckling moment for single symmetric wide flange profiles, ref. NS app. 5.2.2.
### D12.A.3.1 Example

**Note:** This is a partial example containing only the information pertaining to the Norwegian steel design code; used at the end of the input file.

```plaintext
* Code check according to NS3472
PARAMETERS
CODE NS3472
BEAM 1.0 ALL
FYLD 340 ALL
MF 1.10 ALL
CY 0.49 MEMB 1
CZ 0.49 MEMB 1
BY 0.9 MEMB 1
BZ 0.7 MEMB 1
SSY 1.1 MEMB 1
SSZ 1.3 MEMB 1
CB 0.9 MEMB 1
RATIO 1.0 ALL
TRACK 9.0 ALL
UNIT KNS METER
LOAD LIST 1
CHECK CODE MEMB 1
FINISH
```

### D12.A.4 Stability Check According to NS 3472

The stability check is based on the assumption that both ends of the member are structural nodes. Buckling lengths and results for member with joints between the structural nodes have to be evaluated in each separate case.

Effects from local buckling or external hydrostatic pressure on pipes and tubes are not included.

The general stability criteria is: (ref. NS 12.3)

#### D12.A.4.1 Buckling

\[ n_{\text{max}} + k_i \times m_i \leq 1 \]

#### D12.A.4.2 Lateral Buckling

\[ \frac{n}{X_i} + k_{LT} \frac{m_i}{X_{LT}} + k_y m_y \leq 1 \]

Where:

- \( i = z, y \)
- \( n_{\text{max}} = n/X_{\text{min}} \)
- \( n = N_d/N_d \)
- \( X_{\text{min}} = \min(X_z, X_y) \)
- \( X_i = N_{kd,i}/N_d \)
\[ k_i = 1 - \mu_i \frac{n_i}{\chi_i} \leq 1.5 \]
\[ \mu_i = i (2 \beta_{Mi} - 4) \leq 0.9 \]
\[ \beta_{Mi} \text{ ref. NS Tab. 12} \]
\[ k_{LT} = 1 - \mu_{LT} \frac{n_i}{\chi_i} \leq 1.0 \]
\[ \mu_{LT} = 0.15(\beta_{M} - 1) \leq 0.9 \]
\[ i = \lambda_i \Lambda_i \]
\[ \lambda_i = L_{ki} / i_i \]
\[ \lambda_i = \pi \frac{E}{f_y} \]
\[ x_i = \frac{1}{\phi + \sqrt{\phi^2 - \chi_i^2}} \]
\[ \varphi = 0.5[1 + \alpha(-0.2) + \varphi^2] \]
\[ \alpha \text{ ref. NS Tab 10 & 11} \]
\[ \chi_{LT} = \frac{1}{\phi_{LT} + \sqrt{\phi_{LT}^2 - \chi_{LT}^2}} \]
\[ \phi_{LT} = 0.5[1 + \alpha(\chi_{LT} - 0.4) + \chi_{LT}^2] \]
\[ \alpha \text{ ref. NS sec. 12.3.4.1} \]
\[ \chi_{LT} = \sqrt{\frac{W_{iz} f_{iz}}{M_{cr}}} \]
\[ M_{cr} = \psi M_{vio} \]
\[ \psi \text{ ref. NS2 A5.5.2 Sect. a - d} \]
\[ M_{vio} = \frac{n}{L} \sqrt{E T_{iz} G T_{iz}} \left[ 1 + \frac{n^2 E C w}{L^2 G T_{iz}} \right] \]

**D12.A.4.3 Determination of \( \beta_z \) and \( \beta_y \)**

The equivalent moment factor \( \beta \) (for \( z \) and \( y \)) is calculated dependant on moment distributions as shown in the following table:

**Table 183: \( \beta \) for different moment distributions**

<table>
<thead>
<tr>
<th>Moment diagram</th>
<th>( \beta_{M}(\phi_{LT}) )</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Moment Diagram" /></td>
<td>( \beta_{M} \psi = 1.8 - 0.7\psi )</td>
</tr>
</tbody>
</table>
### Design

#### Moment diagram

<table>
<thead>
<tr>
<th>Moment diagram</th>
<th>$\beta_M (\beta_{LT})$</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Moment Diagram" /></td>
<td>$\beta_{M0} = 1.3$</td>
</tr>
<tr>
<td></td>
<td>$\beta_{M0} = 1.4$</td>
</tr>
</tbody>
</table>

- $\beta_M = \beta_{M,\psi} + \frac{M_0}{\Delta M} (\beta_{M,0} - \beta_{M,\psi})$
- $M_0 = |M_{max}|$ due to transverse load only
- $\Delta M = |M_{max}|$ if the moment has the same sign
- $\Delta M = |M_{max}| + |M_{min}|$ if the moment changes sign

The user can override the calculated factor with the following parameters:

- $\beta_y = SSY$
- $\beta_z = SSZ$

#### D12.A.4.4 Lateral buckling

The ideal lateral buckling moment is calculated according to NS2 A5.5.2

$$M_{vi} = \psi M_{vio} = \psi L \left( \frac{E}{L} \sqrt{I_y I_x} \right) \left( 1 + \frac{\pi^2}{L^2} \frac{2.6 C_w}{I_x} \right)$$

concern double symmetric cross sections where $y$ is given in NS fig. A5.5.2, (input parameter CB), $L =$ member length for lateral buckling (input parameter UNL), $C_w$ and $I_x$, see section 5.

For single symmetric cross sections, the ideal lateral buckling moment is

$$M_{vix} = \phi \frac{n^2 E I_y}{L^2} \sqrt{\left( \frac{5a}{n^2} + \frac{r_x}{3} - y_s \right)^2 + C^2} - \left( \frac{5a}{n^2} + \frac{r_x}{3} - y_s \right)$$

Where:
\[ C^2 = \frac{C_w + 0.039L}{I_y} L^2 I_T \]

\( \alpha = \) distance from profile CoG to point where the load is acting, assumed to be on top flange.

The \( \varphi \) parameter (ref NS fig. A5.5.2.g) is controlled by the input parameter CB.

Figure 188: \( \psi \)-coefficients for a simple span beam
Figure 189: $\psi$-coefficients for a partially restrained beam
Figure 190: ψ-coefficients for a fully restrained beam
Figure 191: $\psi$-coefficients for the cantilevered beam with single loads and distributed loads. Dashed curves apply load on the surface.

D12.A.4.5 Stability check of pipe members

The stability criteria applied for members with pipe cross section is:

$$IR = \frac{N}{N_{kd}} + \left[ \frac{M_z}{M_d(1 - \frac{N}{N_{Ezd}})} \right]^2 + \left[ \frac{M_y}{M_d(1 - \frac{N}{N_{Eyd}})} \right]^2 \leq 1.0$$

Where:

$$\frac{N}{N_{kd}} = \max \left( \frac{N}{N_{kzd}}, \frac{N}{N_{kyd}} \right)$$

$\overline{M}_z$ and $\overline{M}_y$ are given in NS 5.4.2.

For the print output option TRACK 9, $\theta K_E \equiv 1.0$ and $M_{vd} \equiv M_d$

D12.A.4.6 Angle profiles type RA (reverse angle)

The axial contribution to the total interaction ratio is checked according to the modified EECS-method, see NS A5.4.
The stability criterion is:

\[
IR = \frac{N}{N_{kd}} + \frac{M_y}{M_{yd}}\left(1 - \frac{N}{N_{Eyd}}\right) + \frac{M_z}{M_{zd}}\left(1 - \frac{N}{N_{Ezd}}\right) \leq 1.0
\]

Where:

\[
\frac{N}{N_{kd}} = \max\left(\frac{N}{N_{kzd}}, \frac{N}{N_{kyd}}\right)
\]

\(N_{kyd}\) and \(N_{kzd}\) are found from NS 3472 fig. 5.4Ja C-curve for y- and z-axis, respectively.

For \(\lambda \leq \sqrt{2}\)

\[
\text{eff} = 0.60 + 0.57
\]

For \(\lambda > \sqrt{2}\)

\[
\text{eff} = \frac{\lambda_k}{\bar{\lambda} - \frac{\pi}{2}}
\]

Where:

\[
\bar{\lambda} = \frac{\lambda_k}{\bar{n}} \sqrt{\frac{I_y}{E}}
\]

\[
\lambda_k = \frac{l_k}{i}
\]

\[
i = \sqrt{I/A}
\]

Possible lateral buckling effects and torsional buckling (NS A5.4.5) is not included in the code check. This has to be evaluated by the user separately.

**D12.A.4.7 Stability check of members with tapered section**

Stability of members with tapered cross section is calculated as described in section 3.1. The cross section properties used in the formulae are calculated based on the average profile height. (i.e., \(I_y, I_z\) values are taken from the middle of the member.)

**D12.A.4.8 Lateral buckling for tension members**

When compressive stress caused by large bending moment about strong axis is greater than tension stress from axial tension force, lateral buckling is considered as defined below.

\[
\sigma_a = \frac{N}{A} (+ \text{tension}, - \text{compression})
\]

\[
\sigma_{bz} = \pm \frac{M_z}{W_z}
\]

\[
M_{warp} = | \sigma_a + \sigma_b | W_z \text{ for } \sigma_a + \sigma_b < 0 \text{ (compression)}
\]

\[
IR = \frac{M_{warp}}{M_{vd}} + \frac{M_{y,\max}}{M_{yd}} \leq 1.0
\]

**D12.A.5 Stability Check According to NPD**

**D12.A.5.1 Buckling of pipe members**

Tubular beam-columns subjected to compression and lateral loading or end moments shall be designed in accordance with NPD 3.2.2
\[ \sigma_c y_{mk} + B o_b^* + \sqrt{(B_z o_{bz})^2 + (B_y o_{by})^2} \leq \frac{f_y}{f_m} \]

Where:

- \( \sigma_c = \frac{N}{A} \) = axial compressive stress
- \( \nu_{mk} \) = structural coefficient
- \( B = \) bending amplification factor = \( \frac{1}{1 - \mu} \), \( B \) is taken as the larger of \( B_z \) and \( B_y \)
- \( B_z = \) bending amplification factor about the Z-axis
- \( B_y = \) bending amplification factor about the Y-axis
- \( \mu = \frac{\sigma_c}{f_y} \)
- \( f_E = \frac{2E}{l_k^2} \)
- \( i = \sqrt{I/A} \)
- \( o_b^* = \sigma_c \left( \frac{f_y}{f_m} - 1 \right) \left( 1 - \frac{f_k}{y_m f_E} \right) \)
- \( l_k = k l \)
- \( k = \) effective length factor
- \( f_k = \) characteristic buckling capacity according to NS fig. 5.4.1a, curve A.

**D12.A.5.2 Interaction with local buckling, NPD 3.2.3**

If the below conditions are not satisfied, the yield strength will be replaced with characteristic buckling stress given in NPD 3.4.

a. members subjected to axial compression and external pressure

\[ \frac{d}{\tau} \leq 0.5 \frac{E}{f_y} \]

b. members subjected to axial compression only

\[ \frac{d}{\tau} \leq 0.1 \frac{E}{f_y} \]

**D12.A.5.3 Calculation of buckling resistance of cylinders**

The characteristic buckling resistance is defined in accordance with NPD 3.4.4

\[ f_k = \frac{f_y}{\sqrt{1 + X^2}} \]

Where:

\[ X^2 = \frac{f_y}{\sigma_j} \left( \frac{a_{ao}}{f_{ea}} + \frac{a_{bo}}{f_{eb}} + \frac{a_{p0}}{f_{ep}} + \frac{t}{f_{et}} \right) \]

\[ \sigma_j = \sqrt{(a_a + a_b)^2 - (a_a + a_b)\sigma_p + \sigma_p^2 + 3\tau^2} \]

- \( \sigma_a \geq 0 \) when \( a_{ao} = 0 \)
- \( \sigma_a < 0 \) when \( a_{ao} = -\sigma_a \)
- \( \sigma_b \geq 0 \) when \( a_{bo} = 0 \)
\( \sigma_b < 0 \) when \( \sigma_{b0} = -\sigma_b \)

\( \sigma_p \geq 0 \) when \( \sigma_{p0} = 0 \)

\( \sigma_p < 0 \) when \( \sigma_{p0} = \sigma_p \)

\( \sigma_a \) = design axial stress in the shell due to axial forces (tension positive)

\( \sigma_b \) = design bending stress in the shell due to global bending moment (tension positive)

\( \sigma_p = \sigma_{p0} \) = design circumferential stress in the shell due to external pressure (tension positive)

\( \tau_S \) = design shear stress in the shell due to torsional moments and shear force.

\( f_{ea}, f_{eb}, f_{ep} \) and \( f_{e\iota} \) are the elastic buckling resistances of curved panels or circular cylindrical shells subjected to axial compression forces, global bending moments, lateral pressure, and torsional moments and/or shear forces respectively.

**D12.A.5.4 Elastic buckling resistance for un-stiffened, closed cylinders**

The elastic buckling resistance for un-stiffened closed cylinders according to NPD 3.4.6 is:

\[
f_e = k \frac{n^2 E}{12(1-\nu^2)} \left( \frac{t}{r} \right)^2
\]

where \( k \) is a buckling coefficient dependent on loading condition, aspect ratio, curvature, boundary conditions, and geometrical imperfections. The buckling coefficient is:

\[
k = \psi \sqrt{1 + \left( \frac{p \zeta}{\psi} \right)^2}
\]

The values of \( \psi, \zeta, \) and \( p \) are given in Table 4.1 for the most important loading cases.

**Table 184: Table 4.1 Buckling coefficients for un-stiffened cylindrical shells**

<table>
<thead>
<tr>
<th></th>
<th>( \psi )</th>
<th>( \zeta )</th>
<th>( p )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial or Bending stress</td>
<td>1</td>
<td>0.702 ( Z )</td>
<td>0.5 ( 1 + \frac{r}{150t} )^{-0.5}</td>
</tr>
<tr>
<td>Torsion and shear force</td>
<td>5.34</td>
<td>0.856 ( Z ) ( 0.75 )</td>
<td>0.6</td>
</tr>
<tr>
<td>Lateral pressure</td>
<td>4</td>
<td>1.04 ( Z ) ( 0.75 )</td>
<td>&quot;</td>
</tr>
<tr>
<td>Hyrdostatic pressure</td>
<td>2</td>
<td>1.04 ( Z ) ( 0.5 )</td>
<td>&quot;</td>
</tr>
</tbody>
</table>

The curvature parameter is defined by

\[
Z = \frac{12}{r} \sqrt{1 - \nu^2}
\]

For long shells the elastic buckling resistance against shear stresses is independent of shell length. For cases with:

\[
\frac{1}{r} > 3.85 \sqrt{\frac{r}{t}}
\]

the elastic buckling resistance may be taken as:

\[
f_{ep} = 0.25 E \left( \frac{t}{r} \right)^2
\]
**D12.A.5.5 Stability requirements**

The stability requirement for curved panels and unstiffened cylindrical shells subjected to axial compression or tension, bending, circumferential compression or tension, torsion or shear is given by NPD 3.4.7:

\[ \sigma_j < f_{kd} \]

where the design buckling resistance is

\[ f_{kd} = \frac{f_k}{\gamma_m \gamma_m} \]

**D12.A.5.6 Column buckling, NPD 3.4.9**

For long cylindrical shells it is possible that interaction between shell buckling and overall column buckling may occur because second-order effects of axial compression alter the stress distribution as compared to that calculated from linear theory. It is necessary to take this effect into account in the shell buckling analysis when the reduced slenderness of the cylinder as a column exceeds 0.2 according to NPD 3.4.4.1.

\[ \sigma_b \] shall be increased by an additional compressive stress which may be taken as:

\[ \Delta \sigma = B \sigma_a \left( \frac{f_y}{f_k} - 1 \right) \left( 1 - \frac{f_k}{f_e} \right) + \left( B - 1 \right) \sigma_b \]

Where:

\[ B = \frac{1}{1 - \mu} \]

\[ \bar{x} = \sqrt{f_y \cdot \frac{f_k}{f_e}} \]

\[ f_e = \frac{\pi^2 E}{\lambda^2} \]

\( \lambda = \) slenderness of the cylinder as a column.

B, \( \sigma_a, \sigma_b, \) and \( \mu \) are calculated in accordance with NPD 3.2.2.

**D12.A.6 Yield Check**

The yield check is performed at member ends and at 11 equally spaced intermediate sections along the member length.

At each section the following forces are applied:

- \( F_x \) max. axial force along member
- \( F_y \) actual shear in local y-direction at section
- \( F_z \) actual shear in local z-direction at section
- \( M_x \) max. torsional moment along member
- \( M_y \) actual bending about local y-axis at section
- \( M_z \) actual bending about local z-axis at section

For all profiles other than angle sections absolute values of the stresses are used. For double symmetric profiles there will always be one stress point.

The stresses are calculated in several stress points at each member section. At each stress
point the von Mises stress is checked as follows:

$$\sigma_j = \sqrt{\sigma_{tot}^2 + \sigma_p^2 - \sigma_{tot} \cdot \sigma_p + 3(\tau_x + \tau_y + \tau_z)^2} \leq \frac{f_y}{\gamma_m}$$

Where:

$$\sigma_{tot} = | \sigma_x + \sigma_{by} + \sigma_{bz} |$$

$$\sigma_p$$ stress from hydrostatic pressure.

**D12.A.B.1 Double symmetric wide flange profile**

The von Mises stress is checked at four stress points as shown in figure below.

![Figure 192: Stress points checked for a wide flange section](image)

**Section Properties**

- $A_x, I_x, I_y,$ and $I_z$ are taken from STAAD.Pro database
- $A_y = h \times s$ Applied in STAAD.Pro print option PRINT MEMBER STRESSES

$$A_x = (2/3) \cdot b \cdot t \cdot 2$$

$$\tau_y = F_y/A_y$$

$$\tau_z = F_z/A_z$$

$A_y$ and $A_z$ are not used in the code check.
Stress calculation

General stresses are calculated as:

\[ \sigma = \sigma_x + \sigma_{by} + \sigma_{bz} = \frac{F_x}{A_x} + \frac{M_y}{I_y} z + \frac{M_z}{I_z} y \]

\[ \tau = \tau_x + \tau_y + \tau_z = \frac{M_x}{I_x c} + \frac{V_y T_z}{I_y t} + \frac{V_z T_y}{I_y t} \]

Where the component stresses are calculated as shown in the following table:

**Table 185: Stress calculations at selected stress points for a wide flange section**

<table>
<thead>
<tr>
<th>Point No</th>
<th>(\sigma_x)</th>
<th>(\sigma_{by})</th>
<th>(\sigma_{bz})</th>
<th>(\tau_x)</th>
<th>(\tau_y)</th>
<th>(\tau_z)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(F_x/A_x)</td>
<td>(M_y b/I_y)</td>
<td>(M_z b/I_z)</td>
<td>(M_x/I_x)</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>&quot;</td>
<td>0</td>
<td>&quot;</td>
<td>&quot;</td>
<td>(F_y bh_2/I_z 2t)</td>
<td>(F_z bh_2/I_y 8t)</td>
</tr>
<tr>
<td>3</td>
<td>&quot;</td>
<td>0</td>
<td>(M_z/I_z h_1)</td>
<td>(M_x/I_x s)</td>
<td>(F_y bh_2/I_z s)</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>&quot;</td>
<td>0</td>
<td>0</td>
<td>&quot;</td>
<td>(F_y (bh_2 + 0.5s^2)/I_z s)</td>
<td></td>
</tr>
</tbody>
</table>

In general wide flange profiles are not suitable for large torsional moments. The reported torsional stresses are indicative only. For members with major torsional stresses a separate evaluation has to be carried out. Actual torsional stress distribution is largely dependent on surface curvature at stress point and warping resistance.

**D12.A.6.2 Single symmetric wide flange profile and tapered section**

The von Mises stress is checked at nine stress points as shown in figure below.
Section properties

$A_x, I_x, I_y$, and $I_z$ are taken from STAAD.Pro database, except for tapered sections where these values are calculated for each section checked. (i.e., $I_z, I_y$ values are taken from the middle of the member.)

$A_z = \frac{2}{3} (b \cdot t + b_1 \cdot t_1)$

$C_w = \frac{b^3_t \cdot b_1^3 t_1 (h - t / 2 - t_1 / 2)^2}{12 (b^3_t + b_1^3 t_1)}$  
ref. NS app. C3

Refer to D12.A.6.1 Double symmetric wide flange profile (on page 2064) for equations used in section property calculations.

Stress calculation

Refer to D12.A.6.1 Double symmetric wide flange profile (on page 2064) for equations used in general stress calculations.

Where the component stresses are calculated as shown in the following table:

**Table 186: Stress calculations at selected stress points for a singly symmetric wide flange section**

<table>
<thead>
<tr>
<th>Point No</th>
<th>$\sigma_x$</th>
<th>$\sigma_{by}$</th>
<th>$\sigma_{bz}$</th>
<th>$\tau_x$</th>
<th>$\tau_y$</th>
<th>$\tau_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$\frac{F_x}{A_x}$</td>
<td>$-\frac{M_y \cdot b}{I_y \cdot 2}$</td>
<td>$\frac{M_x \cdot h_2}{I_z}$</td>
<td>$\frac{M_x}{I_x \cdot t}$</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
In general wide flange profiles are not suitable for large torsional moments. The reported torsional stresses are indicative only. For members with major torsional stresses a separate evaluation has to be carried out. Actual torsional stress distribution is largely dependent on surface curvature at stress point and warping resistance.

**D12.A.6.3 Pipe profile**

The von Mises stress is checked in three stress points as shown in figure below.

<table>
<thead>
<tr>
<th>Point No</th>
<th>$\sigma_x$</th>
<th>$\sigma_{by}$</th>
<th>$\sigma_{bz}$</th>
<th>$\tau_x$</th>
<th>$\tau_y$</th>
<th>$\tau_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>&quot;</td>
<td>0</td>
<td>&quot;</td>
<td>&quot;</td>
<td>$\frac{F_y b t (h_1 + t / 2)}{T_z}$</td>
<td>$\frac{F_y b h b^2}{T_y}$</td>
</tr>
<tr>
<td>3</td>
<td>&quot;</td>
<td>$\frac{M_y b}{I_y}$</td>
<td>&quot;</td>
<td>&quot;</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>&quot;</td>
<td>0</td>
<td>$\frac{M_z}{I_z} h_1$</td>
<td>$\frac{M_x}{I_x} s$</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>&quot;</td>
<td>0</td>
<td>0</td>
<td>&quot;</td>
<td>$\frac{F_y b (h_1 + t / 2)}{T_z}$</td>
<td>$0.5 h_1^2 s$</td>
</tr>
<tr>
<td>6</td>
<td>&quot;</td>
<td>0</td>
<td>$-\frac{M_z}{I_z} h_3$</td>
<td>&quot;</td>
<td>$\frac{F_y b t (h_1 + t / 2)}{I_z}$</td>
<td>$0.5 h_1^2 s$</td>
</tr>
<tr>
<td>7</td>
<td>&quot;</td>
<td>$-\frac{M_y b t}{I_y}$</td>
<td>$-\frac{M_z}{I_z} h_4$</td>
<td>$\frac{M_x}{I_x} t_1$</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>&quot;</td>
<td>0</td>
<td>&quot;</td>
<td>&quot;</td>
<td>$\frac{F_y b t t_1 (h_3 + t_1)}{I_z}$</td>
<td>$\frac{F_y b t_1 (h_3 + t_1)}{I_y}$</td>
</tr>
<tr>
<td>9</td>
<td>&quot;</td>
<td>$\frac{M_y b t}{I_y}$</td>
<td>&quot;</td>
<td>&quot;</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
Figure 194: Stress points for a pipe section

Section properties

\[ d = D - 2t \]
\[ r = 0.5 (D-t) \]
\[ a = \tan^{-1} \frac{M_z}{M_y} \]
\[ A_x = \pi/4 \left(D^2 - d^2\right) \]
\[ A_y = A_z = 0.5A_x \]
\[ I_x = 2I_z = \pi/32 \left(D^4 - d^4\right) \]
\[ I_y = I_z = \pi/64 \left(D^4 - d^4\right) \]

**Note:** In the STAAD.Pro analysis package slightly different values are used for \( A_y, A_z \) and \( I_x \), however this has insignificant influence on the force distribution.

\[ A_y = A_z = 0.6A_x \]
\[ I_x = 2\pi R^2 t \]
Stress calculation at selected stress points

<table>
<thead>
<tr>
<th>Point No</th>
<th>$\sigma_x$</th>
<th>$\sigma_{by}$</th>
<th>$\sigma_{bz}$</th>
<th>$\tau_x$</th>
<th>$\tau_y$</th>
<th>$\tau_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$\frac{F_x}{A_x}$</td>
<td>$\frac{M_y}{I_y}t$</td>
<td>0</td>
<td>$\frac{M_T}{I_x}t$</td>
<td>$\frac{F_y}{0.5A_x}$</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>&quot;</td>
<td>0</td>
<td>$\frac{M_y}{I_y}t$</td>
<td>&quot;</td>
<td>0</td>
<td>$\frac{F_y}{0.5A_x}$</td>
</tr>
<tr>
<td>3</td>
<td>&quot;</td>
<td>$\sigma_b$</td>
<td>$\sigma_b$</td>
<td>&quot;</td>
<td>$\tau$</td>
<td>$\tau$</td>
</tr>
</tbody>
</table>

where

$\sigma_b = \frac{\sqrt{M_y^2 + M_z^2}}{I_z}t$

$\tau = \frac{\sqrt{F_y^2 + F_z^2}}{0.5A_x}$

**D12.A.6.4 Tube profile**

Tube sections are rectangular or quadratic hollow uniform profiles. Critical stress is checked at 5 locations as shown in figure below.
Section Properties

$A_x$, $I_x$, $I_y$, and $I_z$ are taken from the STAAD.Pro database.

where

\[
A_y = 2ht, \text{ similar as for wide flange profiles, see Section 5.2}
\]

\[
A_z = 2(2/3)bt, \text{ Ay and Az are not used in code checks}
\]

\[
C_w = \frac{b^2h^2t(h - b)^2}{24(h + b)} \text{ ref. NS app. C3}
\]

Stress calculation at selected stress points

<table>
<thead>
<tr>
<th>Point No</th>
<th>$\sigma_x$</th>
<th>$\sigma_{by}$</th>
<th>$\sigma_{bz}$</th>
<th>$\tau_x$</th>
<th>$\tau_y$</th>
<th>$\tau_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$\frac{P_x}{A_x}$</td>
<td>$\frac{M_y}{I_y}b_2$</td>
<td>0</td>
<td>$\frac{M_x(h - t)(b - t)}{I_x(h + b - 2t)}$</td>
<td>$\frac{F_y}{bth_2 + th_1^2} + \frac{2}{2t}$</td>
<td>0</td>
</tr>
</tbody>
</table>
The general stress formulation is given in sec. 5.2.

**D12.A.6.5 Channel profile**

For channel profiles the von Mises stress is checked at 6 locations as shown in the figure below.
Cross section properties
where

\[ A_x, S_y, S_z, I_x, I_y, \text{ and } I_z \]

are taken from the STAAD.Pro database

\[ A_y = 2ht, \text{ similar as for wide flange profiles} \]

\[ A_z = 2 \times (2/3)bt, \text{ Ay and Az are not used in code checks} \]

\[ e = h - t \]

\[ y = b / 2 \]

\[ C_w = \frac{x^2 y^2 (2xs + 3yt)}{12 (xs + 6yt)}, \text{ ref.[4] tab. 21, case 1} \]

Stress calculations at selected stress points

<table>
<thead>
<tr>
<th>Point No</th>
<th>( \sigma_x )</th>
<th>( \sigma_{by} )</th>
<th>( \sigma_{bz} )</th>
<th>( \tau_x )</th>
<th>( \tau_y )</th>
<th>( \tau_z )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>( \frac{F_x}{A_x} )</td>
<td>( \frac{M_y}{I_y} (b - e) )</td>
<td>( \frac{M_z}{I_z} h )</td>
<td>( \frac{M_x}{I_x} )</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>( \frac{M_z}{I_z} h )</td>
<td>( \frac{M_x}{I_x} )</td>
<td>( \frac{F_y (b - e) h}{I_y} )</td>
<td>( \frac{0.5 (b - e)^2 , t}{t} )</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>( \frac{M_s}{I_y} b_1 )</td>
<td>( \frac{M_z}{I_z} h )</td>
<td>( \frac{M_x}{I_x} )</td>
<td>( \frac{F_y (b - s) h_2}{I_y} )</td>
<td>( \frac{0.5 (b - s) , t}{t} [0.5 (b - s) - e] )</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>( \frac{M_s}{I_y} b_2 )</td>
<td>( \frac{M_z}{I_z} h )</td>
<td>( \frac{M_x}{I_x} )</td>
<td>( \frac{F_y (b - 0.5 s) h}{I_y} )</td>
<td>( \frac{0.5 (b - s) , t}{t} [0.5 (b - s) - e] )</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>( \frac{M_s}{I_y} b_2 )</td>
<td>( \frac{M_z}{I_z} h_1 )</td>
<td>( \frac{M_x}{I_x} )</td>
<td>( \frac{F_y b t}{I_y} )</td>
<td>( \frac{0.5 (b - e)}{s} )</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>( \frac{M_y}{I_y} b_2 )</td>
<td>0</td>
<td>( \frac{M_x}{I_x} )</td>
<td>( \frac{F_y b t h}{I_y} + 0.5 s h_l )</td>
<td>( \frac{s}{s} )</td>
<td></td>
</tr>
</tbody>
</table>

The general stress formulation is given in sec. 5.2.f

**D12.A.6.6 Angle profile type RA (reverse angle)**

For angle profiles the von Mises check is checked at 8 stress points as shown in figure below.
Axes y and z are principal axes.
Axes u and w are local axes.

Cross section properties
where

\[ A_x, I_x, I_y, \]
\[ A_y \]
\[ A_z \]
\[ \tau_y \]

\[ = \text{taken from the STAAD.Pro database} \]
\[ = \frac{2}{3}ht, \text{applied in STAAD.Pro print option} \]
\[ = \frac{2}{3}bt, Az \text{ are not used in code checks} \]
\[ = \frac{F_x}{A_y} \]
\[\tau_z = \frac{F_x}{A_z}\]
\[h_2 = 0.5h_1 + t\]
\[f = h_1e\]
\[d = \frac{th_1h_2 + 0.5t^2b}{A_x}\]
\[g = \frac{t^2h_1 + tb^2}{2A_x}\]
\[I_u = \frac{h_1^3t^3}{12} + htk^2 + \frac{tb^3}{12} + tb\left(\frac{b}{2} - g\right)^2\]
\[I_w = \frac{th_1^3t^2}{12} + h_1f(h_2 - d)^2 + \frac{bt^3}{12} + bt\left(d - \frac{t}{2}\right)^2\]
\[I_{wu} = \frac{(d - \frac{1}{2})t}{2}\left(g^2 - f^2\right) - \frac{kt}{2}\left(e^2 - f^2\right)\]
\[\alpha = 0.5\tan^{-1}\left(\frac{2I_{uw}}{I_u + I_w}\right)\]

Section forces

The section forces from the STAAD.Pro analysis are about the principle axis y and z.

The second moment of area (Ty L TZ):
\[T_y = A Z\]
\[T_z = A Y\]

Stress calculation at selected stress points

<table>
<thead>
<tr>
<th>Point No</th>
<th>(\sigma_x)</th>
<th>(\sigma_{by})</th>
<th>(\sigma_{bz})</th>
<th>(\tau_x)</th>
<th>(\tau_y)</th>
<th>(\tau_z)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(\frac{F_x}{A_x})</td>
<td>(-\frac{M_yZ1}{I_y})</td>
<td>(-\frac{M_zY1}{I_z})</td>
<td>(\frac{M_x}{I_x})</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>&quot;</td>
<td>0</td>
<td>(-\frac{M_yZ2}{I_y})</td>
<td>&quot;</td>
<td>(\frac{F_yT_x}{I_y})</td>
<td>(\frac{F_zT_x}{I_y})</td>
</tr>
<tr>
<td>3</td>
<td>&quot;</td>
<td>(\frac{M_yZ3}{I_y})</td>
<td>(-\frac{M_zY3}{I_z})</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>4</td>
<td>&quot;</td>
<td>(\frac{M_yZ4}{I_y})</td>
<td>(-\frac{M_zY4}{I_z})</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>5</td>
<td>&quot;</td>
<td>(\frac{M_yZ5}{I_y})</td>
<td>(\frac{M_zY5}{I_z})</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>6</td>
<td>&quot;</td>
<td>0</td>
<td>(\frac{M_zY6}{I_z})</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
</tbody>
</table>
An additional torsional moment is calculated based on:

\[ M_T = F_y Z 4 \]
\[ M_T = F_z Y 4 \]

This torsion moment is included in \( M_x \) if \( F_y \) and \( F_z \) exist.

Beta-rotation of equal & unequal legged angles

**Note:** The order of the joint numbers in the member incidence command specifies the direction of the local x-axis.
**D12.A.6.7 Rectangular massive box (prismatic)**

Code check of the general purpose prismatic cross section defined in the STAAD.Pro analysis package is not available. The prismatic section is assumed to be a rectangular massive box and the von Mises stress is checked at 3 locations as shown in figure below.

![Diagram of rectangular massive box](image)

**Note:** Note that "b" may not be much greater than 'h'. If that is the case, define the member with h > b and Beta angle 90° instead.

### Section Properties

where

- $A_x, A_y, A_z, I_x, I_y,$ and $I_z$ are user-specified. Refer to Section 5.19.2 of the Technical Reference help.
- $b, h = ZD$
- $h = YD$
- $C_w = \frac{1}{24} \frac{b(h-b)^2}{h+b} - \frac{h^2 b^2}{2},$ ref.NS app C3.

### General Stress Calculation

\[
\sigma = \sigma_x + \sigma_{by} + \sigma_{bz} = \frac{F_x}{A_x} + \frac{M_y}{I_y} z + \frac{M_z}{I_z} y
\]

\[
\tau = \tau_x + \tau_y + \tau_z = \tau_{x,max} \left( \frac{c}{b} \right)^2 + \frac{V_y}{A_y} + \frac{V_z}{A_z}
\]

\[
\tau_{x,max} = \frac{M_x (1.5h + 0.9b)}{0.5h \times \frac{b^2}{2}}
\]

ref. [4] tab. 20, case 4 at midpoint the largest (i.e, point 2)

Stress calculation at selected stress points
D12.A.7 Tubular Joint Check, NPD 3.5

For pipe members, punching shear capacity is checked in accordance with the NPD sections 3.5.1 to 3.5.2, except 3.5.2.4. The chord is defined as the member with the greater diameter in the joint. If the diameters are the same the program selects the member with the greater thickness of the two. The chord members must be collinear by 5 degrees.

The punching shear run sequence is performed in two steps. The program will first identify all tubular joints and classify them as T type joints (TRACK99). The joints to be checked will be listed in a file specified in the CODE NPD parameter list, below called GEOM1. This file is used as input in the second run. The file is an editable ACSII file saved under the file name given in the CODE NPD parameter. The TRACK parameter is then set to 98 which directs the program to read from the file GEOM1 file and use it as input to the second run, i.e., the joint capacity checking. The program will check the capacity for both chord members entering the joint. The local y and z moments will be transformed into the plane defined by the joint itself and the far end joints of the brace and chord, defined as in- and out-of plane moments.

The ASCII file should be edited to reflect the correct classification of the joints, gap, can or stub dimensions, yield stress and other geometric options if required. The program will not change the brace or chord definition if this is changed or modified in the input file GEOM1. See Appendix A page xx for GEOM1 example file.

Joint classification parameters in the file GEOM1 are:

- KO K joint overlapped
- KG K joint with gap
- TY T or Y joint
- X X joint

Input example for the classification run.

*CLASSIFICATION OF JOINTS, TRACK 99
UNITS MM NEWTON
PARAMETER
CODE NPD GEOM1
FYLD 350 ALL
TRACK 99 ALL
BEAM 1.0 ALL
CHECK CODE ALL

<table>
<thead>
<tr>
<th>Point No</th>
<th>$\sigma_x$</th>
<th>$\sigma_{by}$</th>
<th>$\sigma_{bz}$</th>
<th>$\tau_x$</th>
<th>$\tau_y$</th>
<th>$\tau_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$F_x/A_x$</td>
<td>$M_y/b$</td>
<td>$M_z/h$</td>
<td>$\tau_{x,\text{max}}$</td>
<td>$b^2/(b^2+h^2)$</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>&quot;</td>
<td>$M_y/b$</td>
<td>0</td>
<td>$\tau_{x,\text{max}}$</td>
<td>$F_y/A_y$</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>&quot;</td>
<td>0</td>
<td>$M_z/h$</td>
<td>$\tau_{x,\text{max}}$</td>
<td>$b^2/h^2$</td>
<td>0</td>
</tr>
</tbody>
</table>
Static strength of tubular joints

The basic consideration is the chord strength. The required chord wall thickness shall be determined when the other dimensions are given.

The following symbols are used:

- \( T \) = Cord wall thickness
- \( t \) = Brace wall thickness
- \( R \) = Outer radius of chord
- \( r \) = Outer radius of brace
- \( \Theta \) = Angle between chord and considered brace
- \( D \) = Outer diameter of chord
- \( d \) = Outer diameter of brace
- \( a \) = Gap (clear distance) between considered brace and nearest load-carrying brace measured along chord outer surface
- \( \beta = r/R \)
- \( g = R/T \)
- \( g = a/D \)
- \( f_y \) = Yield stress
- \( Q_f \) = Factor
- \( Q_g \) = See table 6.1
- \( Q_u \) = See table 6.1
- \( Q_{gd} \) = See table 6.1
- \( N \) = Design axial force in brace
- \( M_{IP} \) = Design in-plane bending moment in brace
- \( M_{OP} \) = Design out-of-plane bending moment in brace
- \( N_k \) = Characteristic axial load capacity of brace (as governed by the chord strength)
- \( M_{OPk} \) = Characteristic out-of-plane bending moment capacity of brace (as governed by the chord strength)
- \( \sigma_{ax} \) = Design axial stress in chord
- \( \sigma_{IP} \) = Design in-plane bending stress in chord
- \( \sigma_{OP} \) = Design out-of-plane bending stress in chord

This section gives design formulae for simple tubular joints without overlap and without gussets, diaphragms or stiffeners. Tubular joints in a space frame structure shall satisfy:

\[
N \leq N_k / \gamma_m
\]

Where:

\[
N_k = Q_u Q_f f_y T^2 / \sin \Theta
\]

\( Q_u \) is given in Table 6.1 and \( Q_f \) is a factor to account for the nominal longitudinal stress in the chord.

\[
Q_f = 1.0 - 0.03\gamma A^2
\]

\[
A^2 = \frac{\sigma_{ax}^2 + \sigma_{IP}^2 + \sigma_{OP}^2}{0.64 f_y^2}
\]
Table 187: Values for $Q_u$

<table>
<thead>
<tr>
<th>Type of joint and geometry</th>
<th>Type of load in brace member</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Axial</td>
</tr>
<tr>
<td></td>
<td>In-plane bending</td>
</tr>
<tr>
<td></td>
<td>Out-of-plane bending</td>
</tr>
<tr>
<td>T and Y</td>
<td>2.5 + 19$\beta$</td>
</tr>
<tr>
<td></td>
<td>$5.0\sqrt{\gamma}\beta$</td>
</tr>
<tr>
<td></td>
<td>$3.2/(1-0.81\beta)$</td>
</tr>
<tr>
<td>X</td>
<td>$(2.7 + 13\beta)Q_\beta$</td>
</tr>
<tr>
<td>K</td>
<td>$0.90(2+21\beta)Q_\beta$</td>
</tr>
</tbody>
</table>

For $\beta > 0.6$, $Q_\beta = 0.3/[\beta(1 - 0.833\beta)]$

For $\beta \leq 0.6$, $Q_\beta = 1.0$

For $\gamma \leq 20$, $Q_g = 1.8 - 0.1a/T$

For $\gamma > 20$, $Q_g = 1.8 - 4g$

but in no case shall $Q_g$ be taken as less than 1.0.

When $\beta \geq 0.9$, $Q_f$ is set to 1.0. This is also applicable for moment loading. For cases with tension in the chord, $Q_f$ is set to 1.0. This is also applicable for moment loading.

The brace end moments shall be accounted for in the following cases:

a. Out-of-plane bending moment when $\beta > 0.85$

b. When the brace acts as a cantilever

c. When the rotational stiffness of the connection is considered in the determination of effective buckling length, and $f$ or the structural coefficient $\gamma_{mk} = 1.00$ for the beam-column design of the brace or chord. See Section 3.1.3.

The characteristic capacity of the brace subjected to in-plane bending moment shall be determined by:

$$M_{IPk} = Q_u Q_f \frac{d f_y T^2}{\sin \theta}$$

Where $Q_u$ is given in Table 6.1 and

$$Q_f = 1.0 - 0.045\gamma A^2$$

The characteristic capacity of the brace subjected to out-of-plane bending moment shall be determined by:

$$M_{OPk} = Q_u Q_f \frac{d f_y T^2}{\sin \theta}$$

Where $Q_u$ is given in Table 6.1 and

$$Q_f = 1.0 - 0.021\gamma A^2$$

For combined axial and bending loads in the brace, the following interaction equation should be satisfied:

$$\frac{N}{N_k} + \left(\frac{M_{IP}}{M_{IPk}}\right)^2 + \frac{M_{OP}}{M_{OPk}} \leq \frac{1}{\gamma_m}$$

For overlapping tubular joints without gussets, diaphragms, or stiffeners, the total load component normal to the chord, $N$, shall not exceed
The formula for the capacity of overlapping joints is valid only for K joints, where compression in a brace is essentially balanced by tension in brace(s) in the same side of the joint.
### TRACK no. Description

<table>
<thead>
<tr>
<th>TRACK no.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>32</td>
<td>Prints maximum and minimum member end forces (axial force defines max and min) at member end 2</td>
</tr>
</tbody>
</table>

**D12.A.8.1 Output for member design**

Output example for TRACK 0.0

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMB</td>
<td>Member number</td>
<td>kN</td>
</tr>
<tr>
<td>FX</td>
<td>Axial force in the member (T = tension, C = compression)</td>
<td>kN-m</td>
</tr>
<tr>
<td>MYs</td>
<td>Start moment about the y-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>MYm</td>
<td>Mid moment about the y-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>MYe</td>
<td>End moment about the y-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>MYb</td>
<td>Buckling moment about the y-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>RATIO</td>
<td>Interaction ratio</td>
<td></td>
</tr>
<tr>
<td>LOAD</td>
<td>The critical load case number</td>
<td></td>
</tr>
<tr>
<td>TABLE</td>
<td>Section type (HE, IPE, TUBE, etc.)</td>
<td></td>
</tr>
<tr>
<td>Mzs</td>
<td>Start moment about z-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>Mzm</td>
<td>Mid moment about the z-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>MZe</td>
<td>End moment about the z-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>MZb</td>
<td>Buckling moment about z-axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>COND</td>
<td>Critical condition</td>
<td></td>
</tr>
<tr>
<td>DIST</td>
<td>Distance from the start of the member to the critical section</td>
<td>m</td>
</tr>
</tbody>
</table>

**Note:** Myb and Mzb are the design moments used for max unity ratio.

---

<table>
<thead>
<tr>
<th>MEMB</th>
<th>TABLE</th>
<th>FX</th>
<th>MYs</th>
<th>MYm</th>
<th>MYe</th>
<th>MYb</th>
<th>RATIO</th>
<th>LOAD</th>
<th>COND</th>
<th>DIST</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>FAIL</td>
<td>12.80 C</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>5.00</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIPS40</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>31.9</td>
<td>-15.9</td>
<td>-36.2</td>
<td>36.2</td>
<td>STAB</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

NS3472 (VERSION 06002) UNITS ARE KN AND METE
Output example for TRACK 1.0

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>CURVE St</td>
<td>Buckling curve about the strong axis</td>
<td></td>
</tr>
<tr>
<td>CURVE Wk</td>
<td>Buckling curve about the weak axis</td>
<td></td>
</tr>
<tr>
<td>Beta Z</td>
<td>Buckling length factor about z-axis</td>
<td></td>
</tr>
<tr>
<td>Beta Y</td>
<td>Buckling length factor about y-axis</td>
<td></td>
</tr>
<tr>
<td>FYLD</td>
<td>Allowable yield strength</td>
<td>N/mm²</td>
</tr>
<tr>
<td>Betamz</td>
<td>Equivalent moment factor $\beta_m$ about z-axis</td>
<td></td>
</tr>
<tr>
<td>Betamy</td>
<td>Equivalent moment factor $\beta_m$ about y-axis</td>
<td></td>
</tr>
<tr>
<td>Fak Z</td>
<td>Factor $k$ according to 12.3.4.2 about the z-axis</td>
<td></td>
</tr>
<tr>
<td>Fak Y</td>
<td>Factor $k$ according to 12.3.4.2 about the y-axis</td>
<td></td>
</tr>
<tr>
<td>MYD</td>
<td>Moment capacity about the y axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>MZD</td>
<td>Moment capacity about the z axis</td>
<td>kN-m</td>
</tr>
<tr>
<td>MVD</td>
<td>Lateral buckling moment</td>
<td>kN-m</td>
</tr>
<tr>
<td>IR1</td>
<td>Interaction ratio for buckling without lateral buckling (Cl. 12.3.4.2)</td>
<td></td>
</tr>
<tr>
<td>IR2</td>
<td>Interaction ratio for buckling with lateral buckling (Cl. 12.3.4.2)</td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>------</td>
</tr>
<tr>
<td>VON MISES</td>
<td>Interaction ratio for von Mises</td>
<td></td>
</tr>
</tbody>
</table>

### NS3472 (VERSION 06002)

**UNITS ARE KN AND METE**

<table>
<thead>
<tr>
<th>MEMB</th>
<th>TABLE</th>
<th>MYs</th>
<th>MYm</th>
<th>MYe</th>
<th>MYb</th>
<th>RATIO</th>
<th>LOAD</th>
<th>COND</th>
<th>DIST</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>FAIL</td>
<td>12.80 C</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>5.08</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIPS40</td>
<td>(AISC SECTIONS)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>36.2</td>
<td>36.2 STAB 10.00</td>
<td></td>
</tr>
</tbody>
</table>

---

**CURVE St A Wk A Beta Z 1.00 Beta Y 1.00 FYLD= 235. N/MM2**

Betamz=1.295 BetaAy=1.000 FakZ=1.500 FakY=1.500
MYD = 0.112E+2 KNM MZD = 0.112E+2 KNM MVD = 0.112E+2 KNM
IR1 = 5.076 IR2 = 5.076 VON MISES = 3.251

---

| 2    | PIP60  | 4.02 C | 0.0  | 0.0  | 0.0  | 0.0  | 0.58  | 1    |
|      |       |       | 36.4 | -38.1 | -6.8 | 38.9 STAB 5.83 |

---

**CURVE St A Wk A Beta Z 1.00 Beta Y 1.00 FYLD= 235. N/MM2**

Betamz=1.377 BetaAy=1.000 FakZ=1.021 FakY=1.033
MYD = 0.701E+2 KNM MZD = 0.701E+2 KNM MVD = 0.701E+2 KNM
IR1 = 0.575 IR2 = 0.575 VON MISES = 0.557

---

| 3    | PIPS40 | 26.31 C | 0.0  | 0.0  | 0.0  | 0.0  | 0.78  | 1    |
|      |       |       | 5.1  | 1.3  | 2.5  | 5.1 STAB 0.00 |

---

**CURVE St A Wk A Beta Z 1.00 Beta Y 1.00 FYLD= 235. N/MM2**

Betamz=2.152 BetaAy=1.000 FakZ=0.602 FakY=1.500
MYD = 0.112E+2 KNM MZD = 0.112E+2 KNM MVD = 0.112E+2 KNM
IR1 = 0.784 IR2 = 0.784 VON MISES = 0.510

---

| 4    | FAIL  | 24.20 C | 0.0  | 0.0  | 0.0  | 0.0  | 1.30  | 1    |
|      | PIPS40| (AISC SECTIONS) | | | | | -2.9  | 2.9 STAB 14.14 |

---

**CURVE St A Wk A Beta Z 1.00 Beta Y 1.00 FYLD= 235. N/MM2**

Betamz=1.510 BetaAy=1.000 FakZ=1.500 FakY=1.500
MYD = 0.112E+2 KNM MZD = 0.112E+2 KNM MVD = 0.112E+2 KNM
IR1 = 1.304 IR2 = 1.304 VON MISES = 0.310

Output example for TRACK 2.0
### NS3472 (VERSION 06001)

**UNITS ARE mm AND N**

<table>
<thead>
<tr>
<th>MEMB</th>
<th>TABLE</th>
<th>Sx</th>
<th>Sby</th>
<th>Sbz</th>
<th>Stot</th>
<th>Spnx</th>
<th>Svm</th>
<th>POINT</th>
<th>LOAD</th>
<th>DIST</th>
</tr>
</thead>
<tbody>
<tr>
<td>111</td>
<td></td>
<td>10.87</td>
<td>242.0</td>
<td>.0</td>
<td>252.9</td>
<td>.0</td>
<td>257.90</td>
<td></td>
<td>11</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 600X15</td>
<td>28.9</td>
<td>0</td>
<td>3</td>
<td>0</td>
<td>3</td>
<td>3.46</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>112</td>
<td></td>
<td>13.70</td>
<td>66.0</td>
<td>.0</td>
<td>79.7</td>
<td>.0</td>
<td>82.70</td>
<td>13</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 600X15</td>
<td>12.4</td>
<td>0</td>
<td>3</td>
<td>.3</td>
<td>0</td>
<td>2.83</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>113</td>
<td></td>
<td>.42</td>
<td>78.7</td>
<td>.0</td>
<td>79.1</td>
<td>.0</td>
<td>80.18</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 600X15</td>
<td>7.4</td>
<td>.2</td>
<td>3</td>
<td></td>
<td></td>
<td>3.46</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>114</td>
<td></td>
<td>13.28</td>
<td>118.6</td>
<td>.0</td>
<td>131.9</td>
<td>.0</td>
<td>135.52</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 600X15</td>
<td>17.7</td>
<td>.3</td>
<td></td>
<td></td>
<td></td>
<td>3</td>
<td>2.83</td>
<td></td>
<td></td>
</tr>
<tr>
<td>115</td>
<td></td>
<td>68.71</td>
<td>18.8</td>
<td></td>
<td>87.5</td>
<td>.0</td>
<td>87.66</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>1.0</td>
<td>.0</td>
<td>2.3</td>
<td></td>
<td></td>
<td>0</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>116</td>
<td></td>
<td>66.13</td>
<td>37.9</td>
<td>.0</td>
<td>104.0</td>
<td>.0</td>
<td>105.91</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>1.1</td>
<td>.0</td>
<td>10.3</td>
<td>.0</td>
<td>3</td>
<td>5.13</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>117</td>
<td></td>
<td>63.29</td>
<td>47.5</td>
<td>.0</td>
<td>110.8</td>
<td>.0</td>
<td>110.84</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>1.7</td>
<td>.0</td>
<td>.4</td>
<td></td>
<td></td>
<td>3</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>118</td>
<td></td>
<td>85.64</td>
<td>39.8</td>
<td>.0</td>
<td>125.4</td>
<td>.0</td>
<td>125.43</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>1.2</td>
<td>.0</td>
<td>.2</td>
<td></td>
<td></td>
<td>3</td>
<td>5.20</td>
<td></td>
<td></td>
</tr>
<tr>
<td>119</td>
<td></td>
<td>90.54</td>
<td>21.4</td>
<td>.0</td>
<td>111.9</td>
<td>.0</td>
<td>111.96</td>
<td>14</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>1.0</td>
<td>.0</td>
<td>.6</td>
<td></td>
<td></td>
<td>3</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>120</td>
<td></td>
<td>94.89</td>
<td>24.7</td>
<td>.0</td>
<td>119.6</td>
<td>.0</td>
<td>119.61</td>
<td>14</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>1.2</td>
<td>.0</td>
<td>.3</td>
<td></td>
<td></td>
<td>3</td>
<td>5.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td>121</td>
<td></td>
<td>79.98</td>
<td>43.7</td>
<td>.0</td>
<td>123.6</td>
<td>.0</td>
<td>123.68</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>1.3</td>
<td>.0</td>
<td>.2</td>
<td></td>
<td></td>
<td>3</td>
<td>5.20</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**SYMBOL**  **DESCRIPTION TRACK 2.0**  **UNIT**

- **MEMB**: Member number
- **Sx**: Axial stress in the member (T = tension, C = compression) **N/mm²**
- **Sby**: Stress from moment about y-axis **N/mm²**
- **Sbz**: Stress from moment about z-axis **N/mm²**
- **Stot**: Sum of Sx + Sby + Sbz **N/mm²**
- **Spnx**: Currently not in use
- **Spmn**: Currently not in use
- **Svm**: von Mises stress **N/mm²**
- **Ty**: Stress from shear force in y direction **N/mm²**
- **Tz**: Stress from shear force in z direction **N/mm²**
- **Tto**: Total shear stress used in von Mises calculation **N/mm²**
- **TABLE**: Section type (HE, IPE, TUBE etc.)
- **POINT**: Location in cross section with max von Mises stress
- **LOAD**: Governing load condition
- **DIST**: Distance from the start of the member to the critical section **m**

**Note:**

Do not use TRACK = 2.0 in connection with the SELECT OPTIMIZED or SELECT MEMBER / ALL commands.
Output example for TRACK 3

<table>
<thead>
<tr>
<th>MEMB</th>
<th>FX</th>
<th>MYs</th>
<th>MYm</th>
<th>MYe</th>
<th>MYb</th>
<th>RATIO</th>
<th>LOAD</th>
<th>COND</th>
<th>LOAD</th>
<th>DIST</th>
</tr>
</thead>
<tbody>
<tr>
<td>111</td>
<td>299.78 T</td>
<td>-13.2</td>
<td>-17.5</td>
<td>21.8</td>
<td>18.4</td>
<td>.86</td>
<td>11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>112</td>
<td>377.68 T</td>
<td>-13.8</td>
<td>-11.8</td>
<td>9.8</td>
<td>12.2</td>
<td>.28</td>
<td>13</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>113</td>
<td>11.61 T</td>
<td>-18.4</td>
<td>-16.4</td>
<td>14.3</td>
<td>16.8</td>
<td>.27</td>
<td>11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>114</td>
<td>366.02 C</td>
<td>-13.2</td>
<td>-11.3</td>
<td>9.3</td>
<td>11.6</td>
<td>.45</td>
<td>11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>115</td>
<td>331.57 C</td>
<td>-4.1</td>
<td>3</td>
<td>-2.6</td>
<td>1.7</td>
<td>.33</td>
<td>11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>116</td>
<td>319.14 T</td>
<td>-1.9</td>
<td>-6.6</td>
<td>-3.3</td>
<td>1.4</td>
<td>.35</td>
<td>11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>117</td>
<td>305.43 T</td>
<td>-4.4</td>
<td>3.7</td>
<td>-7.8</td>
<td>4.5</td>
<td>VMIS 5.13</td>
<td>2087</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>118</td>
<td>413.31 C</td>
<td>10.6</td>
<td>-8.8</td>
<td>9.9</td>
<td>5.3</td>
<td>VMIS 0.00</td>
<td>2087</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>119</td>
<td>436.04 C</td>
<td>-4.5</td>
<td>-1.1</td>
<td>8.8</td>
<td>4.6</td>
<td>STAB 6.20</td>
<td>2087</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>120</td>
<td>457.92 T</td>
<td>-1.6</td>
<td>-2.1</td>
<td>-1.7</td>
<td>1.9</td>
<td>.41</td>
<td>14</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>121</td>
<td>385.99 C</td>
<td>-1.7</td>
<td>-2.1</td>
<td>2.1</td>
<td>1.8</td>
<td>VMIS 5.23</td>
<td>2087</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PIP 200X8</td>
<td>6.0</td>
<td>-8.8</td>
<td>9.7</td>
<td>5.0</td>
<td>STAB 6.20</td>
<td>2087</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**SYMBOl**

<table>
<thead>
<tr>
<th>MEMB</th>
<th>DESCRIPTION TRACK 3.0</th>
<th>UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>Axial force in the member (T= tension, C=compression)</td>
<td>kN</td>
</tr>
<tr>
<td>MYs</td>
<td>Start moment about y-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>MYm</td>
<td>Mid moment about y-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>MYe</td>
<td>End moment about y-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>MYb</td>
<td>Buckling moment about y-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>RATIO</td>
<td>Interaction ratio</td>
<td></td>
</tr>
<tr>
<td>LOAD</td>
<td>The critical load case number</td>
<td></td>
</tr>
<tr>
<td>TABLE</td>
<td>Section type (HE, IPE, TUBE etc.)</td>
<td></td>
</tr>
<tr>
<td>MZs</td>
<td>Start moment about z-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>MZm</td>
<td>Mid moment about z-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>MZe</td>
<td>End moment about z-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>MZb</td>
<td>Buckling moment about z-axis</td>
<td>kNm</td>
</tr>
<tr>
<td>COND</td>
<td>Critical condition</td>
<td></td>
</tr>
<tr>
<td>DIST</td>
<td>Distance from the start of the member to the critical section</td>
<td>m</td>
</tr>
</tbody>
</table>

*TRACK 3: Member results sorted by member number.*
Output example for TRACK 9.0

Member in tension:

DETAILS FOR CODECHECK ACCORDING TO MS3472

(Versions 06001)

MEMBER NO : 111
MEMBER TYPE : PIPE
GOVERNING LOADCASE : 11

<table>
<thead>
<tr>
<th>MEMBER PROPERTY</th>
<th>UNITS</th>
<th>CM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ax</td>
<td>275.7</td>
<td></td>
</tr>
<tr>
<td>Ay</td>
<td>137.8</td>
<td></td>
</tr>
<tr>
<td>Az</td>
<td>137.8</td>
<td></td>
</tr>
<tr>
<td>Ix</td>
<td>236010.1</td>
<td></td>
</tr>
<tr>
<td>Iy</td>
<td>118005.0</td>
<td></td>
</tr>
<tr>
<td>Iz</td>
<td>118005.0</td>
<td></td>
</tr>
</tbody>
</table>

MATERIAL DATA

<table>
<thead>
<tr>
<th>E</th>
<th>204960.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fy</td>
<td>344.966</td>
</tr>
<tr>
<td>Fd</td>
<td>299.870</td>
</tr>
</tbody>
</table>

FORCES

<table>
<thead>
<tr>
<th>Fx</th>
<th>299.78 T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Msx</td>
<td>-426.86</td>
</tr>
<tr>
<td>Msz</td>
<td>262.48</td>
</tr>
<tr>
<td>Msz</td>
<td>-951.82</td>
</tr>
</tbody>
</table>

LATERAL BUCKLING

<table>
<thead>
<tr>
<th>Mlatbuck</th>
<th>670.43</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nvd</td>
<td>1179.90</td>
</tr>
<tr>
<td>IXtot</td>
<td>.786</td>
</tr>
</tbody>
</table>

YIELD CHECK

STRESS : NEW MMS
STRESS AT POINT : 3

<table>
<thead>
<tr>
<th>sigmaax</th>
<th>10.873</th>
</tr>
</thead>
<tbody>
<tr>
<td>sigma</td>
<td>242.016</td>
</tr>
<tr>
<td>tau</td>
<td>28.914</td>
</tr>
<tr>
<td>tors</td>
<td>.305</td>
</tr>
<tr>
<td>sige</td>
<td>257.904</td>
</tr>
<tr>
<td>ITq</td>
<td>.860</td>
</tr>
</tbody>
</table>

Governing interaction ratio .860

Member in compression:
## Design

D. Design Codes

### DETAILS FOR CHECKING ACCORDING TO NS3472 (VERSION 06001)

| MEMBER NO | 50 |
| MEMBER TYPE | TUBE |
| SECTION | TUB16016016 |
| GOVERNING LOADCASE | 5 |

<table>
<thead>
<tr>
<th>MEMBER PROPERTY</th>
<th>UNITS CM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rx</td>
<td>57.4</td>
</tr>
<tr>
<td>Ay</td>
<td>32.0</td>
</tr>
<tr>
<td>Az</td>
<td>32.0</td>
</tr>
<tr>
<td>Ix</td>
<td>3470.0</td>
</tr>
<tr>
<td>Iy</td>
<td>2100.0</td>
</tr>
<tr>
<td>Iz</td>
<td>2100.0</td>
</tr>
</tbody>
</table>

**MATERIAL DATA**

<table>
<thead>
<tr>
<th></th>
<th>UNITS NEWTON MMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>2050000.</td>
</tr>
<tr>
<td>Fy</td>
<td>355.000</td>
</tr>
<tr>
<td>lamfy</td>
<td>15.494</td>
</tr>
<tr>
<td>Gamma</td>
<td>1.100</td>
</tr>
<tr>
<td>Fd</td>
<td>322.727</td>
</tr>
<tr>
<td>n/gamma</td>
<td>1.000</td>
</tr>
</tbody>
</table>

**BUCKLING PARAMETERS**

<table>
<thead>
<tr>
<th></th>
<th>UNITS KNEWTON METERS</th>
</tr>
</thead>
<tbody>
<tr>
<td>L</td>
<td>6.000</td>
</tr>
<tr>
<td>lambda</td>
<td>99.197</td>
</tr>
<tr>
<td>lambda</td>
<td>1.314</td>
</tr>
<tr>
<td>ksi</td>
<td>0.463</td>
</tr>
<tr>
<td>ksi</td>
<td>0.463</td>
</tr>
<tr>
<td>n/ksi</td>
<td>0.032</td>
</tr>
<tr>
<td>n/ksi</td>
<td>0.032</td>
</tr>
<tr>
<td>betaM</td>
<td>2.292</td>
</tr>
<tr>
<td>betaM</td>
<td>2.305</td>
</tr>
<tr>
<td>k</td>
<td>0.978</td>
</tr>
<tr>
<td>k</td>
<td>0.978</td>
</tr>
<tr>
<td>m</td>
<td>0.014</td>
</tr>
<tr>
<td>m</td>
<td>0.066</td>
</tr>
</tbody>
</table>

### FORCES

<table>
<thead>
<tr>
<th></th>
<th>UNITS KNEWTON METERS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fx</td>
<td>27.013 C</td>
</tr>
<tr>
<td>Ms</td>
<td>0.814</td>
</tr>
<tr>
<td>Mn</td>
<td>-0.173</td>
</tr>
<tr>
<td>Me</td>
<td>1.159</td>
</tr>
<tr>
<td>psi</td>
<td>-0.702</td>
</tr>
<tr>
<td>Mmax</td>
<td>1.159</td>
</tr>
<tr>
<td>IRx</td>
<td>0.032</td>
</tr>
<tr>
<td>IRm</td>
<td>0.078</td>
</tr>
<tr>
<td>IKtot</td>
<td>0.110</td>
</tr>
</tbody>
</table>

**YIELD CHECK**

| STRESS | NEW MMS | FORCES | KNEW METERS |
|------------------|------------------|------------------|
| STRESS AT POINT | 3 | FORCES AT SECTION | 6.000 |
| sigmax | 4.706 | Fx | 27.013 C |
| siggo | 24.257 | Fy | -0.329 |
| tau | 0.521 | Fz | 1.616 |
| tors | 0.203 | Mx | 0.094 |
| sigs | 28.990 | My | 5.633 |
| IR | 0.090 | Mz | 1.159 |

**Governing interaction ratio** | 0.110
Member in compression (pipe - NPD):
### DETAILS FOR CODECHECK ACCORDING TO NPD94

(VERSION 96016.00)

**MEMBER NO**: 1  
**MEMBER TYPE**: PIPE 762x 19 mm  
**GOVERNING LOADCASE**: 2

**UNITS**: (cm) [stresses: newtons] [forces: kilonewtons]

**PROPERTIES**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>D/t</td>
<td>40.0</td>
</tr>
<tr>
<td>Iy Iz</td>
<td>306983.8</td>
</tr>
<tr>
<td>Ly</td>
<td>3074.8</td>
</tr>
<tr>
<td>Ax</td>
<td>444.6</td>
</tr>
<tr>
<td>Sy Sz</td>
<td>8057.3</td>
</tr>
<tr>
<td>Lz</td>
<td>3074.8</td>
</tr>
<tr>
<td>Ay Az</td>
<td>222.3</td>
</tr>
<tr>
<td>iy iz</td>
<td>26.3</td>
</tr>
<tr>
<td>By</td>
<td>0.7</td>
</tr>
<tr>
<td>Ix</td>
<td>124263.4</td>
</tr>
<tr>
<td>Z</td>
<td>124263.4</td>
</tr>
<tr>
<td>Bz</td>
<td>0.6</td>
</tr>
</tbody>
</table>

**MATERIAL**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>209979</td>
</tr>
<tr>
<td>Fy</td>
<td>365</td>
</tr>
<tr>
<td>Gamma m</td>
<td>1.150</td>
</tr>
<tr>
<td>lamf y</td>
<td>91.</td>
</tr>
<tr>
<td>Fd</td>
<td>318</td>
</tr>
<tr>
<td>Gamma mk</td>
<td>1.000</td>
</tr>
</tbody>
</table>

**SHELL BUCKLING**

**Section npd 3.4.6.1**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fqk</td>
<td>365.952</td>
</tr>
</tbody>
</table>

**Section npd 3.4.7**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sigj</td>
<td>278.7</td>
</tr>
<tr>
<td>SECT</td>
<td>1.0</td>
</tr>
<tr>
<td>Irshell</td>
<td>0.876</td>
</tr>
</tbody>
</table>

**Section npd 3.4.9.2**

**Bendingmoment stress in 3.4.1.1 increased by dsigb**: 91.430

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>La</td>
<td>0.350</td>
</tr>
<tr>
<td>Lb</td>
<td>1.047</td>
</tr>
<tr>
<td>Fa</td>
<td>249.6</td>
</tr>
</tbody>
</table>

**Section npd 3.4.1**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>lbz</td>
<td>67.872</td>
</tr>
<tr>
<td>lbh</td>
<td>78.404</td>
</tr>
</tbody>
</table>

**Section npd 3.2.2**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIGa</td>
<td>162.6</td>
</tr>
<tr>
<td>SIGby</td>
<td>8.2</td>
</tr>
<tr>
<td>SIGbz</td>
<td>32.2</td>
</tr>
</tbody>
</table>

**Section npd 3.2.2.1 (ns3472 5.4.2)**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fx</td>
<td>7231.114</td>
</tr>
<tr>
<td>Ms</td>
<td>-87.503</td>
</tr>
<tr>
<td>Mm</td>
<td>-154.416</td>
</tr>
<tr>
<td>Me</td>
<td>-297.903</td>
</tr>
<tr>
<td>Beta</td>
<td>-0.294</td>
</tr>
<tr>
<td>m</td>
<td>0.482</td>
</tr>
<tr>
<td>Mb</td>
<td>259.566</td>
</tr>
</tbody>
</table>

**Section npd 3.1.2**

**Stress at point**: 3  
**Forces at section**: 30.748

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>sigax</td>
<td>162.614</td>
</tr>
<tr>
<td>sigb</td>
<td>38.519</td>
</tr>
<tr>
<td>tau</td>
<td>2.156</td>
</tr>
<tr>
<td>tors</td>
<td>0.000</td>
</tr>
<tr>
<td>sigp</td>
<td>30.166</td>
</tr>
<tr>
<td>sige</td>
<td>187.912</td>
</tr>
<tr>
<td>Irj fy</td>
<td>0.590</td>
</tr>
<tr>
<td>HSpres</td>
<td>1.508</td>
</tr>
</tbody>
</table>

**Governing interaction ratio**: 1.227

---

STAAD.Pro 2091  
User Manual
### Design Codes

**D12.A.8.2 Output for Joint Capacity Code Checking**

Output example for TRACK 99.0

<table>
<thead>
<tr>
<th>$JOPNT$</th>
<th>$BRC$</th>
<th>$CHORD$</th>
<th>$D$</th>
<th>$r$</th>
<th>$d$</th>
<th>$t$</th>
<th>$GAP$</th>
<th>$FYC$</th>
<th>$FYD$</th>
<th>$THETA$</th>
<th>$TW$</th>
<th>$THETAT$</th>
<th>$JTYPE$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1020</td>
<td>1016</td>
<td>1015</td>
<td>420.</td>
<td>15.</td>
<td>400.</td>
<td>15.</td>
<td>0.</td>
<td>340.</td>
<td>340.</td>
<td>90.</td>
<td>0.</td>
<td>0.</td>
<td>TY</td>
</tr>
<tr>
<td>1020</td>
<td>1016</td>
<td>1017</td>
<td>420.</td>
<td>15.</td>
<td>400.</td>
<td>15.</td>
<td>0.</td>
<td>340.</td>
<td>340.</td>
<td>90.</td>
<td>0.</td>
<td>0.</td>
<td>TY</td>
</tr>
<tr>
<td>1020</td>
<td>1016</td>
<td>1010</td>
<td>500.</td>
<td>20.</td>
<td>400.</td>
<td>15.</td>
<td>0.</td>
<td>340.</td>
<td>340.</td>
<td>45.</td>
<td>0.</td>
<td>0.</td>
<td>TY</td>
</tr>
<tr>
<td>2011</td>
<td>1016</td>
<td>2010</td>
<td>500.</td>
<td>20.</td>
<td>400.</td>
<td>15.</td>
<td>0.</td>
<td>340.</td>
<td>340.</td>
<td>45.</td>
<td>0.</td>
<td>0.</td>
<td>TY</td>
</tr>
<tr>
<td>2000</td>
<td>1017</td>
<td>1000</td>
<td>500.</td>
<td>20.</td>
<td>420.</td>
<td>15.</td>
<td>0.</td>
<td>340.</td>
<td>340.</td>
<td>45.</td>
<td>0.</td>
<td>0.</td>
<td>TY</td>
</tr>
</tbody>
</table>

Output example for TRACK 98.0

---

**STAAD.Pro User Manual**
Design
D. Design Codes

STAAD.Pro

2093

User Manual


D12.A.8.3 Special prints (not code check)

Output example for TRACK 49

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>MEMBER</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>M1</th>
<th>M2</th>
<th>M3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12</td>
<td>1</td>
<td>46.8</td>
<td>142.0</td>
<td>156.0</td>
<td>72.3</td>
<td>38.8</td>
<td>-31.7</td>
</tr>
<tr>
<td>2</td>
<td>12</td>
<td>1</td>
<td>46.8</td>
<td>-140.7</td>
<td>-156.0</td>
<td>-72.3</td>
<td>88.8</td>
<td>-83.9</td>
</tr>
<tr>
<td>3</td>
<td>12</td>
<td>1</td>
<td>110.8</td>
<td>22.7</td>
<td>-1.1</td>
<td>-1.2</td>
<td>-11.8</td>
<td>82.0</td>
</tr>
<tr>
<td>4</td>
<td>12</td>
<td>1</td>
<td>110.8</td>
<td>22.2</td>
<td>.1</td>
<td>.2</td>
<td>11.4</td>
<td>-82.8</td>
</tr>
<tr>
<td>5</td>
<td>12</td>
<td>1</td>
<td>47.3</td>
<td>-140.1</td>
<td>-158.7</td>
<td>-68.3</td>
<td>-86.7</td>
<td>84.2</td>
</tr>
<tr>
<td>6</td>
<td>12</td>
<td>1</td>
<td>158.5</td>
<td>317.8</td>
<td>63.5</td>
<td>-1.4</td>
<td>75.3</td>
<td>-69.0</td>
</tr>
<tr>
<td>7</td>
<td>12</td>
<td>1</td>
<td>47.3</td>
<td>141.3</td>
<td>158.7</td>
<td>68.3</td>
<td>-43.1</td>
<td>30.9</td>
</tr>
<tr>
<td>8</td>
<td>12</td>
<td>1</td>
<td>19.1</td>
<td>51.6</td>
<td>13.0</td>
<td>-4.4</td>
<td>26.9</td>
<td>-307.4</td>
</tr>
<tr>
<td>9</td>
<td>12</td>
<td>1</td>
<td>192.9</td>
<td>-34.3</td>
<td>-139.6</td>
<td>-16.2</td>
<td>-175.6</td>
<td>-26.5</td>
</tr>
<tr>
<td>10</td>
<td>12</td>
<td>1</td>
<td>19.1</td>
<td>-44.8</td>
<td>-13.0</td>
<td>4.4</td>
<td>30.3</td>
<td>-104.8</td>
</tr>
<tr>
<td>11</td>
<td>12</td>
<td>1</td>
<td>12.1</td>
<td>47.4</td>
<td>119.8</td>
<td>-69.3</td>
<td>43.3</td>
<td>-168.0</td>
</tr>
<tr>
<td>12</td>
<td>12</td>
<td>1</td>
<td>92.2</td>
<td>-118.9</td>
<td>-139.9</td>
<td>-33.0</td>
<td>-174.1</td>
<td>12.4</td>
</tr>
<tr>
<td>13</td>
<td>12</td>
<td>1</td>
<td>2.1</td>
<td>-48.6</td>
<td>-119.8</td>
<td>69.3</td>
<td>54.7</td>
<td>-22.4</td>
</tr>
<tr>
<td>14</td>
<td>12</td>
<td>1</td>
<td>64.5</td>
<td>-22.6</td>
<td>272.9</td>
<td>279.2</td>
<td>151.5</td>
<td>-20.5</td>
</tr>
<tr>
<td>15</td>
<td>12</td>
<td>1</td>
<td>158.5</td>
<td>-71.2</td>
<td>-63.5</td>
<td>1.4</td>
<td>204.0</td>
<td>-347.8</td>
</tr>
<tr>
<td>16</td>
<td>12</td>
<td>1</td>
<td>64.5</td>
<td>-22.3</td>
<td>279.4</td>
<td>279.2</td>
<td>-151.5</td>
<td>-20.5</td>
</tr>
<tr>
<td>17</td>
<td>12</td>
<td>1</td>
<td>7.1</td>
<td>-49.1</td>
<td>-123.3</td>
<td>65.1</td>
<td>-53.0</td>
<td>22.4</td>
</tr>
<tr>
<td>18</td>
<td>12</td>
<td>1</td>
<td>156.1</td>
<td>-73.4</td>
<td>64.0</td>
<td>-1.9</td>
<td>-204.5</td>
<td>-344.3</td>
</tr>
<tr>
<td>19</td>
<td>12</td>
<td>1</td>
<td>9.5</td>
<td>-44.8</td>
<td>13.0</td>
<td>-4.4</td>
<td>-29.2</td>
<td>-302.8</td>
</tr>
<tr>
<td>20</td>
<td>12</td>
<td>1</td>
<td>92.5</td>
<td>12.4</td>
<td>-132.8</td>
<td>18.7</td>
<td>-157.8</td>
<td>-12.8</td>
</tr>
<tr>
<td>21</td>
<td>12</td>
<td>1</td>
<td>218.4</td>
<td>81.6</td>
<td>27.7</td>
<td>75.1</td>
<td>2.3</td>
<td>-94.9</td>
</tr>
<tr>
<td>22</td>
<td>12</td>
<td>1</td>
<td>101.5</td>
<td>46.6</td>
<td>-3.1</td>
<td>-7.8</td>
<td>-9.7</td>
<td>-201.8</td>
</tr>
<tr>
<td>23</td>
<td>12</td>
<td>1</td>
<td>200.7</td>
<td>-33.8</td>
<td>146.5</td>
<td>-10.6</td>
<td>-201.5</td>
<td>-155.7</td>
</tr>
<tr>
<td>24</td>
<td>12</td>
<td>1</td>
<td>549.0</td>
<td>-169.9</td>
<td>3.6</td>
<td>3.4</td>
<td>20.5</td>
<td>223.7</td>
</tr>
<tr>
<td>25</td>
<td>12</td>
<td>1</td>
<td>333.3</td>
<td>.3</td>
<td>2.2</td>
<td>-.2</td>
<td>3.9</td>
<td>-1.0</td>
</tr>
<tr>
<td>26</td>
<td>12</td>
<td>1</td>
<td>40.2</td>
<td>1.1</td>
<td>.9</td>
<td>.1</td>
<td>.1</td>
<td>-2.5</td>
</tr>
</tbody>
</table>

Output example for TRACK 31
D12.B. Norwegian Codes - Steel Design per NORSOK N-004

STAAD.Pro is capable of performing steel design based on the Norwegian code NORSOK N-004 Rev 2, October 2004. Code checks for tubular (pipe) members is performed per the code.

Please note the following:

- The code check is available for the pipe cross sections only.
- The design of conical transitions and joints with joint cans is not performed.

Design of members per NTC 1987 requires the STAAD ECC Super Code SELECT Code Pack.

### Table 1

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MZ</th>
<th>MY</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11</td>
<td>67.5C</td>
<td>153.8</td>
<td>155.2</td>
<td>71.9</td>
<td>38.2</td>
<td>-61.3</td>
</tr>
<tr>
<td>16</td>
<td>-116.3T</td>
<td>60.0</td>
<td>-3.7</td>
<td>-4.2</td>
<td>1.2</td>
<td>-138.9</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>67.5C</td>
<td>153.8</td>
<td>155.2</td>
<td>71.9</td>
<td>38.2</td>
<td>-61.3</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-9C</td>
<td>1.1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-1</td>
</tr>
<tr>
<td>13</td>
<td>48.0C</td>
<td>142.0</td>
<td>156.0</td>
<td>72.3</td>
<td>38.8</td>
<td>-31.6</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-116.3T</td>
<td>60.0</td>
<td>-3.7</td>
<td>-4.2</td>
<td>1.2</td>
<td>-138.9</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>48.0C</td>
<td>142.0</td>
<td>156.0</td>
<td>72.3</td>
<td>38.8</td>
<td>-31.6</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>3.0C</td>
<td>36.3</td>
<td>3</td>
<td>-2</td>
<td>-1</td>
<td>9.0</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>1.1C</td>
<td>96.6</td>
<td>77.1</td>
<td>41.7</td>
<td>39.2</td>
<td>-58.8</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-12.4T</td>
<td>29.1</td>
<td>45.2</td>
<td>24.7</td>
<td>-23.1</td>
<td>30.2</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-81.9T</td>
<td>7.5</td>
<td>-2.6</td>
<td>1.1</td>
<td>-6</td>
<td>103.1</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-116.3T</td>
<td>60.0</td>
<td>-3.7</td>
<td>-4.2</td>
<td>1.2</td>
<td>-138.9</td>
<td></td>
</tr>
</tbody>
</table>

### Table 2

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MZ</th>
<th>MY</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11</td>
<td>67.6C</td>
<td>-152.5</td>
<td>-155.2</td>
<td>-71.9</td>
<td>88.7</td>
<td>-64.0</td>
</tr>
<tr>
<td>16</td>
<td>-116.3T</td>
<td>-58.7</td>
<td>3.7</td>
<td>-1</td>
<td>-4.2</td>
<td>133.4</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-9C</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>11</td>
<td>67.6C</td>
<td>-152.5</td>
<td>-155.2</td>
<td>-71.9</td>
<td>88.7</td>
<td>-64.0</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-116.3T</td>
<td>-58.7</td>
<td>3.7</td>
<td>-1</td>
<td>-4.2</td>
<td>133.4</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>48.0C</td>
<td>-140.8</td>
<td>-156.0</td>
<td>-72.3</td>
<td>88.8</td>
<td>-84.0</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>3.0C</td>
<td>-35.4</td>
<td>-3</td>
<td>2</td>
<td>-2</td>
<td>20.3</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>48.0C</td>
<td>-140.8</td>
<td>-156.0</td>
<td>-72.3</td>
<td>88.8</td>
<td>-84.0</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>14.7C</td>
<td>-56.0</td>
<td>-91.8</td>
<td>-42.8</td>
<td>-52.3</td>
<td>31.6</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-116.3T</td>
<td>-58.7</td>
<td>3.7</td>
<td>-1</td>
<td>-4.2</td>
<td>133.4</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-81.9T</td>
<td>-7.5</td>
<td>2.6</td>
<td>-1</td>
<td>2.7</td>
<td>-97.0</td>
<td></td>
</tr>
</tbody>
</table>
D12.B.1 Member Resistances

The implementation of the NORSOK N-004 code in STAAD.Pro considers sections 4, 5, 6 & 7 in of that document. The details of the various clauses implemented from these sections is presented here for member checking and design.


The general safety check is per Section 4. Checks are made to ensure that the design action effect ($S_d$) is less than or equal to the design resistance ($R_d$):

$$S_d \leq R_d$$

The design resistance is evaluated for each condition and this check is applied as described in the following sections.

D12.B.1.2 Steel selection and non destructive testing

Section 5 deals with the choice of “design class” for structural joints and components. The choice of design class will determine the choice of steel grade & quality and also the determination of inspection category for fatigue. The choice of design class (as per Table 5-1 of the code) is left to you and does not have any direct impact on how STAAD.Pro performs design checks.

D12.B.1.3 Ultimate Limit States

Clause 6.1 primarily deals with the section of material factors to be used in the various conditions or checks. The material factors chosen are dependent on the “section class” of a cross section. N-004 does not explicitly specify how to classify various cross sections. Therefore, the section classification is made as given in Section 5.5 of EN 1993-1-1:2005, except when specified explicitly along with member checks (See Member Subject to Axial Compression).

Also, N-004 does not specify steel grades to be used. Therefore, this STAAD.Pro uses the steel grades per EN 1993-1-1:2005 for designs per N-004.

Note: Ring stiffener design to CL. 6.3.6.2 is not included for this implementation.

D12.B.1.4 Tubular Members

Clause 6.3.1 deals with the general considerations while using tubular members.

Caution: Only tubular sections can be used with the N-004 code in STAAD.Pro. A warning is presented for any other section type.

The dimensions of the tubular sections are limited as follows:

- The thickness $t \geq 6$ mm.
- The thickness $t < 150$ mm.
- The slenderness ratio of the cross section $D/t < 120$.

Where $D$ is the diameter and $t$ is the wall thickness of the section.

- The yield strength for tubular member $\leq 500$ N/mm$^2$.

If any of these conditions are not met for a member selected for design, a warning will be issued by the engine and the design of that member is aborted.
**Note:** N-004 uses "Y" to define the action effects that is in plane and 'Z' to define out of plane effects. This is the opposite to what STAAD uses, where 'Z' defines the in plane effects and 'Y' the out of plane effects. This document will follow the STAAD.Pro convention for the Z and Y axes.

The N-004 code also segregates members into those that are subject to hydrostatic pressure and those that are not subject to hydrostatic pressure. The program allows you to specify whether a member is subject to hydrostatic pressure or not and, if so, to specify the hydrostatic pressure for the element. By default the program will assume that all members are not subject to any hydrostatic pressure. The design parameter HYD is used to specify the maximum water level with respect to the origin.

If the HYD parameter is specified, the program will take that to be the water level and will evaluate the pressure distribution on each element assuming a linear increase in pressure with depth (The density of water is assumed to be 9.8 KN/m$^3$). Also, if the HYD parameter is specified, the program will assume that the hydrostatic loads have not been included in the analysis. For members that are subject to a combination of loads (i.e., bending plus compression) along with a hydrostatic pressure, the design will be done according to Clause 6.3.9 of the code. In the absence of any hydrostatic pressure on the member the design will be performed in accordance with Clause 6.3.8 of the code.

**D12.B.1.5 Ultimate Limit State**

**Axial Tension**

Clause 6.3.2 states that tubular members subject to axial tension shall satisfy the following condition:

\[
N_{Sd} \leq N_{t,Rd} = A \cdot f_y / \gamma_m
\]

where

- $N_{Sd}$ = Design axial force (tension positive)
- $f_y$ = Characteristic yield strength
- $A$ = Cross section area
- $\gamma_m$ = Default material factor = 1.15

**Axial Compression**

Clause 6.3.3 states that tubular members subject to axial compression shall satisfy the following condition:

\[
N_{Sd} \leq N_{c,Rd} = A \cdot f_c / \gamma_m
\]

where

- $N_{Sd}$ = Design axial force (compression positive)
- $f_c$ = Characteristic axial compressive strength
- $\gamma_m$ = Refer to clause 6.3.7

The design axial compressive strength for a member that is not subject to any hydrostatic pressure will be taken as the smaller of in plane or out of plane buckling strengths determined by the equations given below:

\[
f_c = [1.0 - 0.28 \cdot 2] f_y \text{ when } \leq 1.34
\]

\[
f_c = 0.9/2 \cdot f_y \text{ when } > 1.34
\]

\[
= \sqrt{(f_{cl}/f_E)} = k \cdot l/(\pi \cdot i) \cdot \sqrt{(f_{cl}/E)}
\]

where

- $f_{cl}$ = Characteristic local buckling strength
- $\lambda$ = Column slenderness parameter
- $f_E$ = Smaller Euler buckling strength in y or z direction
- $E$ = Young's modulus of elasticity = 2.1x10$^5$ MPa
- $k$ = Effective length factor, refer to Clause 6.3.8.2
The characteristic local buckling strength is determined from:

\[ f_{cl} = f_y \text{ when } f_y/f_{cle} \leq 0.170 \text{ (Plastic yielding)} \]
\[ f_{cl} = (1.047 \cdot 0.274 \cdot f_y/f_{cle}) \cdot f_y \text{ when } 0.170 < f_y/f_{cle} \leq 1.911 \text{ (Elastic/Plastic)} \]
\[ f_{cl} = f_{cle} \text{ when } f_y/f_{cle} > 1.911 \text{ (Elastic buckling)} \]

where

\[ f_{cle} = 2C_e E t / D \text{ (Characteristic elastic local buckling strength)} \]
\[ C_e = 0.3 \text{ (Critical elastic buckling coefficient)} \]
\[ D = \text{Outside diameter} \]
\[ t = \text{wall thickness} \]

For a member that is subject to pure compression, if \( f_y/f_{cle} > 0.170 \), the section will be classed as a CLASS 4 (slender section). In such cases, the value of the material factor \( (\gamma_m) \) used in the above checks is increased according to equation 6.22 (Cl. 6.3.7) of the code.

**Bending**

Clause 6.3.4 states that tubular members subject to pure bending alone shall satisfy:

\[ M_{Sd} \leq M_{Rd} = f_m \cdot W / \gamma_m \]

where

\( M_{Sd} \) = Design bending moment
\( f_m \) = Characteristic bending strength
\( W \) = Elastic section modulus
\( \gamma_m \) = Refer to clause 6.3.7

The bending strength \( f_m \) is calculated as:

\[ f_m = Z/W \cdot f_y \text{ when } f_y D/(E \cdot t) \leq 0.0517 \]
\[ f_m = [1.13 \cdot 2.58 \cdot f_y D/(E \cdot t)] \cdot Z/W \cdot f_y \text{ when } 0.0517 < f_y D/(E \cdot t) \leq 0.1034 \]
\[ f_m = [0.94 \cdot 0.76 \cdot f_y D/(E \cdot t)] \cdot Z/W \cdot f_y \text{ when } 0.1034 < f_y D/(E \cdot t) \leq 120 \cdot f_y / E \]

**Shear**

Clause 6.3.5 states that tubular members subject to shear shall satisfy:

\[ V_{Sd} \leq V_{Rd} = A \cdot f_y / (2 \sqrt{3} \cdot \gamma_m) \]

where

\( V_{Sd} \) = Design shear force
\( f_y \) = Yield strength
\( A \) = Cross section area
\( \gamma_m \) = Default material factor = 1.15

When torsional shear stresses are present, the following condition shall also be satisfied:

\[ M_{T,Sd} \leq M_{T,Rd} = 2 \cdot I_p f_y / (D \sqrt{3} \cdot \gamma_m) \]

where

\( M_{T,Sd} \) = Design bending moment
\( I_p \) = Polar moment of inertia

**Hydrostatic Pressure**
Clause 6.3.6 states that tubular members subject to an external pressure shall primarily be checked for hoop buckling. The condition to be satisfied is:

\[ \sigma_{p,Sd} \leq \frac{f_h}{\gamma_m} \]

where

- \( \sigma_{p,Sd} = \frac{p_{Sd} \cdot D}{2 \cdot t} \)
- \( p_{Sd} \) = Design hydrostatic pressure
- \( f_h \) = Characteristic hoop buckling strength
- \( \gamma_m \) = Refer to clause 6.3.7

The characteristic hoop buckling strength \( f_h \) will be calculated as follows:

- \( f_h = f_y \) when \( f_{he} > 2.44 \cdot f_y \)
- \( f_h = 0.7 \cdot f_y \left( \frac{f_{he}}{f_y} \right) ^ {0.4} \) when \( 2.44 \cdot f_y \geq f_{he} > 0.55 \cdot f_y \)
- \( f_h = f_{he} \) when \( f_{he} \leq 0.55 \cdot f_y \)

where

The elastic hoop buckling strength, \( f_{he} \), is evaluated as follows:

\[ f_{he} = 2C_h E \cdot t / D \]

where

- \( C_h = 0.44 \cdot t / D \) when \( \mu \geq 1.6 \cdot D / t \)
- \( C_h = 0.44 \cdot t / D + 0.21 \cdot (D / t)^3 / \mu^4 \) when \( 0.825 \cdot D / t \leq \mu < 1.6 \cdot D / t \)
- \( C_h = 0.737 / (\mu - 0.579) \) when \( 1.5 \leq \mu < 0.825 \cdot D / t \)
- \( C_h = 0.8 \) when \( \mu < 1.5 \)
- \( \mu = \) Geometric Parameter = \( L / D \sqrt{(2 \cdot D / t)} \)
- \( L = \) Length of tubular member between stiffening rings, diaphragms, or end connections.

Combined Axial Tension and Bending (without Hydrostatic Pressure)

Clause 6.3.8.1 states that tubular members subject to axial tension and bending shall be designed to satisfy the following condition:

\[ \left( \frac{N_{Sd}}{N_{t,Rd}} \right) ^{1.75} + \frac{\sqrt{M_{y,Sd}^2 + M_{z,Sd}^2}}{M_{Rd}} \leq 1.0 \]

where

- \( M_{y,Sd} \) = the design bending moment about the y axis (out-of plane axis)
- \( M_{z,Sd} \) = the design bending moment about the z axis (in plane axis)
- \( N_{Sd} \) = the design axial force
- \( M_{Rd} \) = the moment resistance (as determined by Clause 6.3.4)
- \( N_{t,Rd} \) = the tension capacity of the section (as determined by Clause 6.3.2)

Combined Axial Compression and Bending (without Hydrostatic Pressure)

Clause 6.3.8.2 states that tubular members subject to axial tension and bending shall be designed to satisfy the following conditions:

\[ \frac{N_{Sd}}{N_{c,Rd}} + \frac{1}{M_{Rd}} \left[ \frac{C_{my} M_{y,Sd}^2}{N_{Sd} N_{Ey}} \right] ^2 + \frac{C_{mz} M_{z,Sd}^2}{N_{Sd} N_{Ez}} \leq 1.0 \]

and
\[
\frac{N_{Sd}}{N_{cl,Rd}} + \sqrt{\frac{M_{y,Sd}^2 + M_{z,Sd}^2}{M_{Rd}}} \leq 1.0
\]

where

\begin{align*}
N_{Sd} & = \text{the design axial compression} \\
C_{my} \text{ and } C_{mz} & = \text{the reduction factors corresponding to the Y and Z axes, respectively. You may specify a value for these using the CMY and CMZ design parameters, respectively (default 0.85 for both).} \\
N_{ey} \text{ and } N_{ez} & = \text{the Euler buckling loads about y & z axes and are given by:} \\
N_{Ey} & = \frac{n^2EA}{k l / y} \\
N_{Ez} & = \frac{n^2EA}{k l / z} \\
k & = \text{the effective length factor and is given in table 6-2 of the code.} \\
N_{cl,Rd} & = \text{the design axial local buckling resistance given by:} \\
N_{cl,Rd} & = \frac{f_{cl}A}{\gamma_M} \\
f_{cl} & = \text{the characteristic local buckling strength (as determined by Clause 6.3.3)}
\end{align*}

The reduction factors used in this clause depend on the “structural element type” and will be as given in Table 6-2 of N-004. This requires the member to be classified under any one of the section types given in the table.

**Combined Bending and Shear (without Hydrostatic Pressure)**

Clauses 6.3.8.3 & 6.3.8.4 state that tubular members subject to beam shear force (excluding shear due to torsion) and bending moments shall satisfy:

\[
\frac{M_{Sd}}{M_{Rd}} \leq \sqrt{1.4 - \frac{V_{Sd}}{V_{Rd}}} \text{ when } \frac{V_{Sd}}{V_{Rd}} \geq 0.4
\]

\[
\frac{M_{Sd}}{M_{Rd}} \leq 1.0 \text{ when } \frac{V_{Sd}}{V_{Rd}} < 0.4
\]

If the member is subject to shear forces due to torsion along with bending moments, the condition to be satisfied is:

\[
\frac{M_{Sd}}{M_{Red,Rd}} \leq \sqrt{1.4 - \frac{V_{Sd}}{V_{Rd}}} \text{ when } \frac{V_{Sd}}{V_{Rd}} \geq 0.4
\]

\[
\frac{M_{Sd}}{M_{Red,Rd}} \leq 1.0 \text{ when } \frac{V_{Sd}}{V_{Rd}} < 0.4
\]

where

\begin{align*}
M_{Red,Rd} & = W \cdot f_{m,Red}/\gamma_m \\
f_{m,Red} & = f_m \sqrt{[1 - 3(\tau_{T,Sd}/f_d)^2]} \\
\tau_{T,Sd} & = M_{T,Sd}/(2\pi \cdot R^2 - t) \\
f_{cl} & = f_y/\gamma_m \\
R & = \text{Radius of the tubular member} \\
\gamma_m & = \text{Refer to clause 6.3.7}
\end{align*}

**Combined Loads with Hydrostatic Pressure**

Clause 6.3.9 of NS-004 describes two methods to check for members subject to combined forces in the presence of hydrostatic pressure: depending on whether the hydrostatic forces were included as nodal forces in the analysis or not. If the hydrostatic forces have not been included in the analysis as nodal forces, Method A given in the code is used. If, however, the hydrostatic forces have been included in the analysis, then Method B in the
code is used. Prior to proceeding with the checks described in the sections below, the section is verified for hoop stress limit per clause 6.3.6 (see Hydrostatic Pressure above).

The choice of method for checking members subject to combined forces and hydrostatic pressure used by STAAD.Pro will depend on the HYD parameter specified as a design parameter. If the HYD parameter has been specified, then the program will assume that the hydrostatic forces have not been included in the analysis and will perform the necessary checks as per Method A in code. If, on the other hand, the HYD parameter has not been specified, the program will use the section forces and use Method B in the code.

Combined Axial Tension, Bending, and Hydrostatic Pressure
Checks per Clause 6.3.9.1:
A. When HYD is specified:

The following condition is to be satisfied:

a. For the net axial tension condition \( (\sigma_{asd} \geq \sigma_{q,asd}) \)

\[
\frac{\sigma_{a,asd} - \sigma_{q,asd}}{f_{th,Rd}} + \frac{\sqrt{\sigma_{my,asd}^2 + \sigma_{mz,asd}^2}}{f_{mh,Rd}} \leq 1.0
\]

where

\( \sigma_{a,asd} \) = the design axial stress, excluding any axial compression from hydrostatic pressure.

\( \sigma_{q,asd} \) = the design axial compressive stress due to hydrostatic pressure. (i.e., the axial load arising from the hydrostatic pressure being applied as nodal loads).

\( \sigma_{my,asd} \) = the out of plane bending stress

\( \sigma_{mz,asd} \) = the in plane bending stress

\( f_{th,Rd} \) = \( f_y / \gamma_m \sqrt{1 + 0.09 \cdot B^2 - B^2 \eta} - 0.3 \cdot B \)

\( f_{mh,Rd} \) = \( f_m / \gamma_m \sqrt{1 + 0.09 \cdot B^2 - B^2 \eta} - 0.3 \cdot B \)

\( B \) = \( \sigma_{psd} / f_{h,Rd} \)

\( \eta \) = \( 5 - 4 \cdot f_{he} / f_y \)

b. For the net axial compression condition \( (\sigma_{asd} < \sigma_{q,asd}) \)

\[
\frac{|\sigma_{a,asd} - \sigma_{q,asd}|}{f_{cl,Rd}} + \frac{\sqrt{\sigma_{my,asd}^2 + \sigma_{mz,asd}^2}}{f_{mh,Rd}} \leq 1.0
\]

where

\( f_{cl,Rd} \) = \( f_{cl} / \gamma_m \)

\( f_{cl} \) = the characteristic local buckling strength (as determined by Clause 6.3.3)

Additionally, when:

\( \sigma_{c,asd} > 0.5 \cdot f_{he} / \gamma_m \)

and

\( f_{cle} > 0.5 \cdot f_{he} \)

the following condition shall be satisfied in addition to the above check(s):
Design

D. Design Codes

\[
\sigma_{c,Sd} - 0.5 \frac{f_{he}}{\gamma_M} + \left( \frac{\sigma_{p,Sd}}{f_{he}} \right)^2 \leq 1.0
\]

where

\[
\sigma_{c,Sd} = \text{the maximum compressive stress at that section.}
\]

B. When HYD has not been specified:

\[
\frac{\sigma_{ac,Sd}}{f_{th,Rd}} + \sqrt{\frac{\sigma_{my,Sd}}{f_{my,Rd}}^2 + \frac{\sigma_{mz,Sd}}{f_{mz,Rd}}^2} \leq 1.0
\]

where

\[
\sigma_{ac,Sd} = \text{the axial stress in the member}
\]

Combined Axial Compression, Bending, and Hydrostatic Pressure

Checks per Clause 6.3.9.2:

A. Method used when HYD has been specified:

The following condition is to be satisfied:

\[
\frac{\sigma_{a,Sd}}{f_{ch,Rd}} + \frac{1}{f_{mh,Rd}} \sqrt{\left( \frac{C_{my} \sigma_{my,Sd}}{f_{EY}} \right)^2 + \left( \frac{C_{mz} \sigma_{mz,Sd}}{f_{Ez}} \right)^2} \leq 1.0
\]

and

\[
\frac{\sigma_{a,Sd} + \sigma_{q,Sd}}{f_{cl,Rd}} + \sqrt{\frac{\sigma_{my,Sd}}{f_{my,Rd}}^2 + \frac{\sigma_{mz,Sd}}{f_{mz,Rd}}^2} \leq 1.0
\]

Where:

where

\[
\sigma_{a,Sd} = \text{the design axial stress that excludes the stress from hydrostatic pressure}
\]

\[
f_{ch,Rd} = \frac{1}{2} \frac{f_{cl}}{\gamma_M} \left( 1 - \frac{2 \sigma_{q,Sd}}{f_{cl}} \right) \sqrt{2 \sigma_{q,Sd} f_{cl}^2} \text{ when } \lambda < 1.34 \left( 1 - \frac{2 \sigma_{q,Sd}}{f_{cl}} \right)^{-1}
\]

\[
\xi = \frac{0.9 f_{cl}}{\lambda^2 \gamma_M} \text{ when } \lambda \geq 1.34 \left( 1 - \frac{2 \sigma_{q,Sd}}{f_{cl}} \right)^{-1}
\]

Additionally, when:

\[
\sigma_{c,Sd} > 0.5 \cdot f_{he}/\gamma_m
\]

and

\[
f_{cle} > 0.5 \cdot f_{he}
\]

the following condition shall be satisfied in addition to the above check(s):
\[
\frac{\sigma_{c,Sd} - 0.5 \cdot f_{he}}{\gamma_M} + \left( \frac{\sigma_{p,Sd}}{f_{he}} \right) \leq 1.0
\]

B. Method used when HYD has not been specified:

The following condition is to be satisfied:

a. For the net axial tension condition \((\sigma_{ac,Sd} \geq \sigma_{q,Sd})\)

\[
\frac{\sigma_{ac,Sd} - \sigma_{q,Sd}}{f_{ch,Rd}} + \frac{1}{f_{mh,Rd}} \left[ \frac{C_{my} \sigma_{my,Sd}}{1 - \frac{\sigma_{a,Sd}}{f_{E_y}}} \right]^2 + \frac{C_{mz} \sigma_{mz,Sd}}{1 - \frac{\sigma_{a,Sd}}{f_{E_z}}} \leq 1.0
\]

and

\[
\frac{\sigma_{ac,Sd}}{f_{cl,Rd}} + \frac{\sqrt{\sigma_{my,Sd}^2 + \sigma_{mz,Sd}^2}}{f_{mh,Rd}} \leq 1.0
\]

(Refer to the previous section for an explanation of these terms).

b. For the net axial compression condition \((\sigma_{ac,Sd} < \sigma_{q,Sd})\)

\[
\frac{\sigma_{ac,Sd}}{f_{cl,Rd}} + \frac{\sqrt{\sigma_{my,Sd}^2 + \sigma_{mz,Sd}^2}}{f_{mh,Rd}} \leq 1.0
\]

(Refer to the previous section for an explanation of these terms).

Additionally, when:

\[\sigma_{c,Sd} > 0.5 \cdot f_{he}/\gamma_m\]

and

\[f_{cle}/\gamma_m > 0.5 \cdot f_{he}/\gamma_m\]

the following condition shall be satisfied in addition to the above check(s):

\[
\frac{\sigma_{c,Sd} - 0.5 \cdot f_{he}}{\gamma_M} + \left( \frac{\sigma_{p,Sd}}{f_{he}} \right) \leq 1.0
\]

where

\[\sigma_{c,Sd} = \text{the maximum compressive stress at that section.}\]

D12.B.3 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.
Table 189: Design Parameters for NORSOK N-004 design code

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| CODE           | none          | Must be specified as NORSOK.  
|                |               | **Note**: Do not use the shortened NOR, as this initiates an NS3472 design (on page 2046). |
| FYLD           | 235 [MPa]     | Yield strength of steel, \( f_y \) (St37)  
<p>|                |               | <strong>Note</strong>: Note, if the SGR value is specified, then the associated value of ( f_y ) for that steel grade will be used for a member in lieu of the FYLD value. |
| KY             | 1.0           | Effective length factor, ( k ), in local Y-axis, usually minor axis. |
| KZ             | 1.0           | Effective length factor, ( k ), in local Z-axis, usually major axis. |
| LX             |               | Effective length for lateral torsional buckling. |
| LY             | Member Length | Length in local Y axis for slenderness value ( KL/r ) |
| LZ             | Member Length | Length in local Z axis for slenderness value ( KL/r ) |
| CMY            | 0.85          | Reduction factor ( C_m ) corresponding to the Y axis. |
| CMZ            | 0.85          | Reduction factor ( C_m ) corresponding to the Z axis. |
| LSR            |               | Length of Tubular between Stiffening Rings. This value is required to calculate Design Hoop Stress due to Hydrostatic Pressure to check Hoop Buckling as per clause 6.3.6.1. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| HYD            | 0.0           | The Y-coordinate, current units, of the maximum water level with respect to the origin.  
**Note:** If SET Z_UP command has been specified, then yi will be the Z co-ordinate of the max water level.  
For HYD > 0, the value of max. hydrostatic pressure calculated is reported for each member in a TRACK 2.0 output. |
| PSD            | 0.0           | Water pressure at each section in absence of HYD. |
| SGR            | 0.0           | Steel Grade per EC3 ([EN 1993-1-1:2005](#)):
- 0.0 = S 235 grade steel  
- 1.0 = S 275 grade steel  
- 2.0 = S 355 grade steel  
- 3.0 = S 420 grade steel  
- 4.0 = S 460 grade steel |
| DMAX           | 100.0 [cm]    | Maximum allowable depth of steel section. |
| DMIN           | 0.0 [cm]      | Minimum allowable depth of steel section. |
| DFF            | None (Mandatory for deflection check) | "Deflection length"/maximum allowable local deflection. |
| MAIN           | 0.0           | Option to design for slenderness.  
- 0.0 = Check for slenderness  
- 1.0 = Do not check for slenderness  
- Any value greater than 1.0 is used as the limit for slenderness in compression. |
| TMAIN          | 180.0         | Slenderness limit in tension. Slenderness limit is checked based the MAIN parameter. |
| FU             | 420 [MPa]     | Ultimate Tensile Strength of Steel |
### Parameter Name | Default Value | Description
--- | --- | ---
 TRACK | 0.0 | Output detail:
  0.0 = Only a summary of the design checks performed is printed.
  2.0 = All the details of the member checks and the various clause checks performed are printed.
 RATIO | 1.0 | Permissible ratio of the actual to allowable stresses.
 BEAM | 0.0 | Beam segment locations for design:
  0.0 = design only for end moments and those at locations specified by SECTION command.
  1.0 = Perform design for moments at twelfth points along the beam.
 DJ1 | Start Joint of member | Joint No. denoting start point for calculation of “deflection length”
 DJ2 | End Joint of member | Joint No. denoting end point for calculation of “deflection length”

### Notes
**a. C1 and C2 Parameters**

The default values of these coefficients are taken from Table 6-4 of N-004 and depend on the joint and load type:

**Table 190: Default values for C1 and C2 parameters**

<table>
<thead>
<tr>
<th>Joint Type</th>
<th>C1</th>
<th>C2</th>
</tr>
</thead>
<tbody>
<tr>
<td>T or Y joints under brace axial load</td>
<td>25</td>
<td>11</td>
</tr>
<tr>
<td>X joints under brace axial load</td>
<td>20</td>
<td>22</td>
</tr>
<tr>
<td>K joints under balanced axial load</td>
<td>20</td>
<td>22</td>
</tr>
<tr>
<td>All joints under brace moment loading</td>
<td>25</td>
<td>30</td>
</tr>
</tbody>
</table>
**D12.B.4 Code Checking**

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate as per N-004. Code checking is done using the forces and moments at specific sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks, the critical condition of NORSOK code, the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value), the governing load case, and the location (distance from the start of the number of forces in the member) where the critical condition occurs.

**D12.B.5 Member Selection**

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section (i.e., the lightest section which fulfills the code requirements for the specified member). The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN which limit the maximum and minimum depth of the members.

Selection of members whose properties are originally input from a user created table will be limited to sections in the user table.

**D12.B.6 Tubular Joint Checking**

The design of tubular joints for this implementation shall be based on section 6.4 of N-004 and will be applicable to joints formed from a connection of two or more members.
Prior to completing a joint design, the joint should be classified into one of the three categories given by the code. Joint classification is the process whereby a BRACE member connecting into a CHORD member is classified into one of these categories based on the axial force components in the brace. The classification normally considers all the members at a joint that lie in a plane. N-004 defines three joint classification categories: K, X, or Y (or a combination of these).

<table>
<thead>
<tr>
<th>Joint Classification</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>K</td>
<td>The axial force in the brace should be balanced by forces in the other braces in the same plane and on the same side of the joint. The code allows a 10% tolerance in the balancing force.</td>
</tr>
<tr>
<td>X</td>
<td>The axial force in the brace is reacted as a beam shear in the chord.</td>
</tr>
<tr>
<td>Y</td>
<td>The axial force in the brace is carried through the chord to braces in the opposite side.</td>
</tr>
</tbody>
</table>

**Note:** Typical examples of these joint types are given in Figure 6-3 of the N-004 code. It is worth noting that the joint class for each brace will be different for each load case.

**Note:** STAAD.Pro does not perform an automatic classification of the joints. This is left up to the engineer. All joints will initially be classified as Y in the generation of the external geometry file (on page 2110). Joints should be re-classified as necessary before performing the final joint capacity checks.

The checks for joint capacity are given in Cl. 6.4.3.2 to 6.4.3.6 and STAAD.Pro performs the checks as per these clauses. However, the program does not deal with conical joint transitions and joints with joint cans. The code also specifies checks and limits for the gaps and eccentricity of joints. This implementation will not perform such geometry checks.

The details of the checks done and the methodology will be discussed in the following sections.
D12.B.6.1 Identification and Classification of CHORD and BRACE Members

This is a two step process where the program automatically identifies the CHORD and BRACE members at a joint and perform a default joint check. The input variables used for the initial joint checks will be generated in an external text file. You can then use this text file to edit or modify the input variables and perform a final check as necessary.

The following syntax is used to initiate the joint checking in the engine.

```
LOAD LIST load_list
PARAMETER 1
CHECK JOINT { node_list | ALL }
```

Where:
- `load_list` = a list of load case numbers to be check against
- `node_list` = the NODE numbers to be checked. Specifying the ALL keyword option will cause the program to perform the joint check at all the nodes.

For each node specified in the CHECK JOINT command, the program automatically separates out all the members at the node into one CHORD member and one or more BRACE members. The section with the biggest diameter is assumed to be the CHORD and all the other members are assumed as BRACE members. If two or more possible CHORD members have the same diameter, the member with the maximum thickness is considered as the CHORD. The angle between the two members should be within the range of 30° and 90° (inclusive).

Once all the CHORD and BRACE members are identified, the program considers every CHORD to BRACE connection as a separate JOINT. The program then performs all the necessary joint checks as detailed in the following sections and produces the design output. The program will also produce an output file called `filename_JOINTS.txt`, where "filename" will be the name of the .std file. This format of this text file is explained in D12.B.8 External Geometry File (on page 2110).

You can then edit this text file to set up the necessary design parameters. Once the program finds of the _JOINTS.txt file, it will read in the necessary parameters from this file and perform the subsequent design checks.

**Note:** This file will be produced only once (i.e., when this file does not exist). If this file exists, it is assumed that you have already done a joint design check and hence the program reads the values from this file and uses these for joint checks.

D12.B.7 Tubular Joint Resistance

D12.B.7.1 Basic Joint Resistances

The characteristic joint resistance between a chord and a brace is given by:

\[
N_{Rd} = \frac{f_y T^2}{\gamma_M \sin \theta} Q_u Q_f
\]

\[
M_{Rd} = \frac{f_y T^2 d}{\gamma_M \sin \theta} Q_u Q_f
\]
Where:

\( N_{Rd} \) is the joint design axial resistance
\( M_{Rd} \) is the joint design bending moment resistance.
\( f_y \) is the yield strength
\( \gamma_m = \text{Default material resistance} = 1.15 \)
\( \theta \) is the angle between the chord and the brace (max \( \theta = 90 \) degrees)
\( Q_u = \text{Strength factor which varies with the joint type and the action type in the brace. Refer to Table 6-3 and Clause 6.4.3.3 of N-004 for these equations.} \)
\( Q_f = 1.0 - \lambda A^2 \)

\[ A^2 = C_1 \left( \frac{\sigma_{P,SD}}{f_y} \right)^2 + C_2 \left( \frac{\sigma_{my,SD}^2 + \sigma_{mz,SD}^2}{1.62f_y^2} \right) \]

\( \sigma_{P,SD} \) is the design axial stress in the chord
\( \sigma_{my,SD} \) is the design in-plane bending stress in the chord
\( \sigma_{mz,SD} \) is the design out-of-plane bending stress in the chord
\( C_1 \) is the coefficient used for the axial stress term in calculating the joint resistance. \( C_2 \) is the coefficient used for the bending stress term in calculating the joint resistance. The default values of \( C_1 \) and \( C_2 \) are as given in Table 6-4 of N-004. The actual values used are dependent on the values of \( K, X, \) and \( Y \) specified for the joint in the external geometry file (on page 2110).

See also Figures 6-3 to 6-6 of N-004 for definition of the various terms for various joint classes.

**D12.B.7.2 Strength Check for Joints**

Each brace to chord joint to be checked will have to satisfy the following condition:

\[ \frac{N_{Sd}}{N_{Rd}} + \left( \frac{M_{z,SD}}{M_{z,Rd}} \right)^2 + \frac{M_{y,SD}}{M_{y,Rd}} \leq 1 \]

Where:

\( N_{Sd} \) is the design axial force in the brace,
\( N_{Rd} \) is the joint design axial resistance
\( M_{z,SD} \) is the in plane bending moment in the brace
\( M_{y,SD} \) is the out of plane bending moment in the brace
\( M_{z,Rd} \) is the in plane bending moment resistance
\( M_{y,Rd} \) is the out of plane bending moment resistance

**D12.B.8 External Geometry File**

The data contained in the `filename_JOINTS.NGo` file should meet the following format. The overall process of performing punching shear checks consists of two steps which are explained in [D12.B.6 Tubular Joint Checking](on page 2107).
**General Format**

LOAD LIST Load_List

<table>
<thead>
<tr>
<th>JOINT</th>
<th>NODE</th>
<th>K</th>
<th>X</th>
<th>Y</th>
<th>CHORD</th>
<th>CLEN</th>
<th>D</th>
<th>T</th>
<th>BRACE</th>
<th>BLEN</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>n</td>
<td>K%</td>
<td>X%</td>
<td>Y%</td>
<td>C#</td>
<td>CLEN</td>
<td>D</td>
<td>T</td>
<td>B#</td>
<td>BLEN</td>
</tr>
</tbody>
</table>

Where:

- \( j\# = \) the joint number
- \( n\# = \) the node number
- \( K\%, X\%, \text{and} \ Y\% = \) The fractional contributions of K-type, X type and Y-type, respectively. Initially the joints will be classed as Y (i.e., \( K=0, X=0 \) and \( Y=1 \)).
- \( C\# = \) the member numbers of the CHORD
- \( CLEN = \) the length of chord member
- \( D, T = \) Diameter and thickness of CHORD
- \( B\# = \) the member number of the brace
- \( BLEN = \) the length of chord member
- \( d, t = \) Diameter and thickness of BRACE
- \( \text{gap} = \) Distance required to calculate gap factor for K bracing. Initially, the value of GAP is assumed as 0.

**Example**

<table>
<thead>
<tr>
<th>LOAD LIST 1 2 4</th>
<th>JOINT</th>
<th>NODE</th>
<th>K</th>
<th>X</th>
<th>Y</th>
<th>CHORD</th>
<th>CLEN</th>
<th>D</th>
<th>T</th>
<th>BRACE</th>
<th>BLEN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3</td>
<td>0.140</td>
<td>0.010</td>
<td>1</td>
<td>2</td>
<td>5.0</td>
<td>0.168</td>
<td>0.10</td>
<td>1</td>
<td>4.0</td>
<td></td>
</tr>
<tr>
<td>4.043</td>
<td>0.075</td>
<td>0.005</td>
<td>0</td>
<td>1</td>
<td>2</td>
<td>5.0</td>
<td>0.168</td>
<td>0.10</td>
<td>16</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**D12.B.9 Tabulated Results**

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

- **Member** the member number for which the design is performed.
- **TABLE** the steel section name which has been checked against the N-004 code or has been selected.
- **RESULTS** prints whether the member has PASSed or FAILed. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
- **CRITICAL COND** the section of the N-004 code which governs the design.
- **RATIO** prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
- **LOADING** the load case number which governed the design.
- **FX, MY, and MZ** provide the axial force, moment in local Y-axis, and the moment in local Z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to
perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.

**LOCATION** specifies the actual distance from the start of the member to the section where design forces govern.

**Note:** If the parameter TRACK is set to 2.0, the program will block out part of the table and will print the allowable bending stressed in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV).

---

**D12.B.9.1 Sample TRACK 2.0 Output**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>1</td>
<td>ST</td>
<td>PASS</td>
<td>Eq. 6.44</td>
<td>0.170</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.01 C</td>
<td>1.01</td>
<td>6.39</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**MATERIAL DATA**
- Grade of steel = S 355
- Modulus of elasticity = 204999.98 N/mm²
- Design Strength (py) = 355.00 N/mm²

**SECTION PROPERTIES (units - cm)**
- Member Length = 400.00
- Gross Area of cross section = 40.70
- z-axis
  - Moment of inertia = 862.000
  - Plastic modulus = 168.554
  - Elastic modulus = 123.407
  - Radius of gyration = 4.602
  - Effective Length = 400.000
- y-axis
  - Moment of inertia = 862.000
  - Plastic modulus = 168.554
  - Elastic modulus = 123.407
  - Radius of gyration = 4.602
  - Effective Length = 400.000

**DESIGN PARAMETER (units - m)**
- N004/2004
- Height of water lavel = 3.000
- CMZ = 0.85
- CMY = 0.85
- KZ = 1.00
- KY = 1.00

**SECTION CLASSIFICATION**
- Class 1

**CAPACITIES (units - kN,m)**
- Tension Capacity = 1256.4
- Compression Capacity = 790.1
- Bending Capacity = 52.0
- Shear Capacity = 362.7
- Shear Capacity due to torsional moment = 44.0

**HYDROSTATIC PRESSURE CALCULATION (units - N,mm)**
- Cl.6.3.6
- Max design hydrostatic pressure, (psd) = 0.000
- Max design hoop stress, (sigma_psd) = 0.000
### D13. Russian Codes

#### D13.A. Russian Codes - Concrete Design Per SNiP 2.03.01-84*

STAAD.Pro is capable of performing concrete design based on the Russian code СНиП 2.03.01-84*:

**СТРОИТЕЛЬНЫЕ НОРМЫ И ПРАВИЛА БЕТОННЫЕ И ЖЕЛЕЗОБЕТОННЫЕ КОНСТРУКЦИИ (СНиП 2.03.01-84* Building Regulations: Concrete and Reinforced Concrete Construction).**

Design of members per SNiP 2.03.01-84* requires the **STAAD ECC Super Code SELECT Code Pack.**

#### D13.A.1 General

Russian Code SNiP 2.03.01–84* plain concrete and concrete structures is based on the method of limit states. Code SNiP 2.03.01–84* defines two groups of limit states.

Analysis according to the first group of limit states is performed to avoid the following phenomena:

- brittle, plastic or other type of failure,
- loss by structure of stable form or position,
- fatigue failure,
- failure due to the action of load actions and unfavorable environmental effects.

Analysis according to the second group of limit states is performed to avoid the following phenomena:

- excessive and long-term opening of cracks if they are allowed according to service conditions,
- excessive displacements.

Analysis of structures for the first group of limit states is performed with the use of the maximum (design) loads and actions. Analysis of structures for the second group of limit states is made in accordance with the operational (normative) loads and actions. Ratio between design and normative loads is called reliability coefficient for loads which is determined according to SNiP 2.01.07.-85 "Loads and actions".

Reliability coefficient $\gamma_{n}$ for destination according to SNiP 2.01.07.-85 shall be considered in determination of loads and their combinations.

Program STAAD.Pro makes it possible to calculate reinforcement for concrete members according to codes of many countries round the World and Russian Code SNiP 2.03.01-84* inclusive. Algorithms for calculation of reinforcement of concrete linear (beams, columns) and 2D (two dimensional) (slabs, walls, shells) members are incorporated in program STAAD.Pro. Not only Code SNiP 2.03.01-84* but also the “Guide for design of plain

<table>
<thead>
<tr>
<th>CRITICAL LOAD FOR EACH CLAUSE CHECK (units - kN,m):</th>
<th>CLAUSE</th>
<th>RATIO LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl:6.3.2</td>
<td>0.000</td>
<td>1</td>
<td>0.0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Cl:6.3.3</td>
<td>0.000</td>
<td>1</td>
<td>0.0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Cl:6.3.4</td>
<td>0.102</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-5.3</td>
<td>0.0</td>
</tr>
<tr>
<td>Cl:6.3.5</td>
<td>0.031</td>
<td>1</td>
<td>-</td>
<td>-11.2</td>
<td>0.5</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Cl:6.3.8.(1 &amp; 2)</td>
<td>0.124</td>
<td>1</td>
<td>0.0</td>
<td>-</td>
<td>-</td>
<td>6.4</td>
<td>1.0</td>
</tr>
<tr>
<td>Cl:6.3.8.(3 &amp; 4)</td>
<td>0.102</td>
<td>1</td>
<td>-</td>
<td>-0.5</td>
<td>0.5</td>
<td>-5.3</td>
<td>0.0</td>
</tr>
<tr>
<td>Cl:6.3.9</td>
<td>0.170</td>
<td>1</td>
<td>0.0</td>
<td>-</td>
<td>-</td>
<td>6.4</td>
<td>1.0</td>
</tr>
</tbody>
</table>
concrete and reinforced concrete structures from normal weight and lightweight concrete (to SNiP 2.03.01-84)"
have been used in creation of these algorithms.

It is possible using program STAAD.Pro to calculate reinforcement for beams of rectangular or T section and for
columns of rectangular or circular section (Fig.1).

![Figure 196: Notation of dimensions for rectangular, circular and T sections](image)

Flange of T-shape beams may be situated at the top zone of the section if the angle BETA=0°, or at the bottom
zone of the section, if BETA=180°.

**D13.A.2 Design Parameters**

Entry of data of cross-sections of beams and columns is made by the use of **MEMBER PROPERTIES** command,
and thicknesses of 2D members are entered by **ELEMENT PROPERTY** command.

**Example:**

```
UNIT MM
MEMBER PROPERTIES
* Columns of rectangular cross-section
  1 TO 16 PRI YD 350. ZD 350.
* Columns of circular cross-section
  17 TO 22 PRI YD 350.
* Beams of T cross-section
  23 TO 40 PRI YD 450. ZD 550. YB 230. ZB 200.
```

```
UNIT METER
ELEMENT PROPERTY
41 TO 100 THICKNESS 0.14
101 TO 252 THICKNESS 0.16
* Flange of T beams is located at the bottom zone of cross-section
BETA 180. MEMB 23 TO 40
```

Commands for calculation of reinforcement are located in the input data file after the command of analysis and
as a rule, after output commands to print results of calculation.

**Example:**

```
* Command of analysis
PERFORM ANALYSIS
.
* Output command to print results of calculation (according to user's judgment)
```
In tables 1, 2 and 3 information about parameters used for calculation of reinforcement for beams, columns and 2D (two dimensional) members is presented. Values of parameters do not depend on UNIT command. In the file of input data only such parameters have to be taken, the values of which differ from determined in the program.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 191: Names of parameters for Concrete design according to Russian Code -СНиП 2.03.01-84* for beams.

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>NLT</td>
<td>1</td>
<td>Number of long-term loading case</td>
</tr>
<tr>
<td>No.</td>
<td>Parameter name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-----</td>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| 2   | RCL            | 3             | Class of longitudinal reinforcement:  
|     |                |               | - RCL = 1, if class of reinforcement is A-I;  
|     |                |               | - RCL = 2, if class of reinforcement is A-II;  
|     |                |               | - RCL = 3, if class of reinforcement is A-III;  
|     |                |               | - RCL = 33, if class of reinforcement is A-IIIb;  
|     |                |               | - RCL = 4, if class of reinforcement is A-IV;  
|     |                |               | - RCL = 5, if class of reinforcement is A-V;  
|     |                |               | - RCL = 6, if class of reinforcement is A-VI;  
|     |                |               | - RCL = 7, if class of reinforcement is A-VII;  
|     |                |               | - RCL = 77, if class of reinforcement is K-7;  
|     |                |               | - RCL = 8, if class of reinforcement is B-II;  
|     |                |               | - RCL = 9, if class of reinforcement is Bp-II;  
|     |                |               | - RCL = 10, if class of reinforcement is Bp-I;  
|     |                |               | - RCL = 19, if class of reinforcement is K-19  |

2

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| 2   | RCL            | 3             | Class of longitudinal reinforcement: Russian Grade:  
|     |                |               | - 1 = A240;  
|     |                |               | - 2 = A300;  
|     |                |               | - 3 = A400;  
|     |                |               | - 4 = A500;  
|     |                |               | - 5 = B500;  
|     |                |               | - 6 = A500SP;  
|     |                |               | European Grade:  
|     |                |               | - 11 = S240;  
|     |                |               | - 12 = S400;  
<p>|     |                |               | - 13 = S500;  |</p>
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>USM</td>
<td>1.</td>
<td>Total product of service conditions coefficients for longitudinal reinforcement ( (g_s) )</td>
</tr>
<tr>
<td>4</td>
<td>UB2</td>
<td>0.9</td>
<td>Specific service conditions coefficient for concrete ( (g_{b2}) )</td>
</tr>
<tr>
<td>5</td>
<td>DD1</td>
<td>16.</td>
<td>Diameter of longitudinal reinforcement bars in beam tension zone</td>
</tr>
<tr>
<td>6</td>
<td>DD2</td>
<td>16.</td>
<td>Diameter of shear reinforcement bars for beam;</td>
</tr>
<tr>
<td>7</td>
<td>BCL</td>
<td>15.</td>
<td>Compression class of concrete</td>
</tr>
<tr>
<td>No.</td>
<td>Parameter name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-----</td>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| 7   | BCL            | 15.           | Compression Class of concrete.  
|     |                |               | • 10 = B10;  
|     |                |               | • 15 = B15  
|     |                |               | • 20 = B20;  
|     |                |               | • 25 = B25;  
|     |                |               | • 30 = B30;  
|     |                |               | • 35 = B35;  
|     |                |               | • 40 = B40;  
|     |                |               | • 45 = B45;  
|     |                |               | • 50 = B50;  
|     |                |               | • 55 = B55;  
|     |                |               | • 60 = B60;  
|     |                |               | • 8.10 = C8/10  
|     |                |               | • 12.15 = C12/15;  
|     |                |               | • 16.20 = C16/20  
|     |                |               | • 25.30 = C25/30  
|     |                |               | • 30.37 = C30/37  
|     |                |               | • 35.45 = C35/45  
|     |                |               | • 40.50 = C50/50  
|     |                |               | • 45.55 = C45/55  
|     |                |               | • 50.60 = C50/60  
|     |                |               | • 60.75 = C60/75  
|     |                |               | • 70.85 = C70/85  
|     |                |               | • 80.95 = C80/95  
|     |                |               | • 90.105 = C90/105  
| 8   | UBM            | 1.            | Product of service conditions coefficients for concrete, except UB2 ($b_0$)  
| 9   | TEM            | 0.            | Parameter of concrete hardening conditions:  
|     |                |               | • TEM=0, for natural hardening conditions;  
|     |                |               | • TEM=1, for steam hardening conditions  
| 10  | CL1            | 0.05          | Distance from top/bottom fiber of beam cross section to the center of longitudinal reinforcement bar;  

**Design**  
D. Design Codes

*STAAD.Pro 2118*  
*User Manual*
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>CL2</td>
<td>0.05</td>
<td>Distance from left/right side of beam cross section to the center of longitudinal reinforcement bar</td>
</tr>
<tr>
<td>12</td>
<td>WST</td>
<td>0.4</td>
<td>Ultimate width of short-term crack</td>
</tr>
<tr>
<td>13</td>
<td>WLT</td>
<td>0.3</td>
<td>Ultimate width of long-term crack</td>
</tr>
</tbody>
</table>
| 14  | SSE            | 0             | Limit state parameter for beam design  
  - SSE=0, if calculation of reinforcement amount must be carried out according to the requirements of load carrying capacity (the first limit state);  
  - SSE=1, if calculation of reinforcement amount must be carried out according to the cracking requirements (the second limit state) |
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| 15  | RSH            | 1             | Class of shear reinforcement:  
  - RSH = 1, if class of reinforcement is A-I;  
  - RSH = 2, if class of reinforcement is A-II;  
  - RSH = 3, if class of reinforcement is A-III;  
  - RSH = 33, if class of reinforcement is A-IIIb;  
  - RSH = 4, if class of reinforcement is A-IV;  
  - RSH = 5, if class of reinforcement is A-V;  
  - RSH = 6, if class of reinforcement is A-VI;  
  - RSH = 7, if class of reinforcement is A-VII;  
  - RSH = 77, if class of reinforcement is K-7;  
  - RSH = 8, if class of reinforcement is B-II;  
  - RSH = 9, if class of reinforcement is Bp-II;  
  - RSH = 10, if class of reinforcement is Bp-I;  
  - RSH = 19, if class of reinforcement is K-19  

| 15  | RSH            | 1             | Class of shear reinforcement:  
  Russian Grade:  
  - 1 = A240;  
  - 2 = A300;  
  - 3 = A400;  
  - 4 = A500;  
  - 5 = B500;  
  - 6 = A500SP;  
  European grade:  
  - 11 = S240;  
  - 12 = S400;  
  - 13 = S500;  |
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>FWT</td>
<td>ZD</td>
<td>Design width of beam top flange. Use for beam design only with default value provided as ZD in member properties.</td>
</tr>
<tr>
<td>17</td>
<td>FWB</td>
<td>ZB</td>
<td>Design width of beam bottom flange. Use for beam design only with default value provided as ZB in member properties.</td>
</tr>
<tr>
<td>18</td>
<td>DEP</td>
<td>YD</td>
<td>Design depth of beam section. Use for beam design only with default value provided as YD in member properties.</td>
</tr>
<tr>
<td>19</td>
<td>SFA</td>
<td>0.</td>
<td>Face of support location at the start of the beam. Use for beam design only.</td>
</tr>
<tr>
<td>20</td>
<td>EFA</td>
<td>0.</td>
<td>Face of support location at the end of the beam. Use for beam design only.</td>
</tr>
<tr>
<td>21</td>
<td>NSE</td>
<td>13</td>
<td>Number of equally-spaced sections for beam design. Use for beam design only. Upper limit is equal to 20.</td>
</tr>
</tbody>
</table>

Table 192: Names of parameters for Concrete design according to Russian Code СНиП 2.03.01-84* for columns

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>NLT</td>
<td>1</td>
<td>Number of long-term loading case</td>
</tr>
<tr>
<td>No.</td>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-----</td>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| 2   | RCL            | 3             | Class of longitudinal reinforcement: Russian Grade:  
|     |                |               | • 1 = A240;  
|     |                |               | • 2 = A300;  
|     |                |               | • 3 = A400;  
|     |                |               | • 4 = A500;  
|     |                |               | • 5 = B500;  
|     |                |               | • 6 = A500SP;  
|     |                |               | European Grade:  
|     |                |               | • 11 = S240;  
|     |                |               | • 12 = S400;  
<p>|     |                |               | • 13 = S500;  |
| 3   | USM            | 1.            | Total product of service conditions coefficients for longitudinal reinforcement (g_s) |
| 4   | UB2            | 0.9           | Specific service conditions coefficient for concrete (g_{b2}) |
| 5   | DD1            | 16.           | Minimum diameter of longitudinal reinforcement bars for column |
| 6   | DD2            | 16.           | Maximum diameter of longitudinal reinforcement bars for column |</p>
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| 7   | BCL            | 15            | Compression class of concrete:  
• 10 = B10;  
• 15 = B15  
• 20 = B20;  
• 25 = B25;  
• 30 = B30;  
• 35 = B35;  
• 40 = B40;  
• 45 = B45;  
• 50 = B50;  
• 55 = B55;  
• 60 = B60;  
• 8.10 = C8/10  
• 12.15 = C12/15;  
• 16.20 = C16/20  
• 25.30 = C25/30  
• 30.37 = C30/37  
• 35.45 = C35/45  
• 40.50 = C50/50  
• 45.55 = C45/55  
• 50.60 = C50/60  
• 60.75 = C60/75  
• 70.85 = C70/85  
• 80.95 = C80/95  
• 90.105 = C90/105 |
| 8   | UBM            | 1.            | Product of service conditions coefficients for concrete, except UB2 (Bb) |
| 9   | TEM            | 0.            | Parameter of concrete hardening conditions:  
• TEM=0, for natural hardening conditions;  
• TEM=1, for steam hardening conditions |
| 10  | CL1            | 0.05          | Distance from edge of column cross section to the center of longitudinal reinforcement bar |
### Table 193: Names of parameters for Concrete design according to Russian Code (SNiP 2.03.01-84*) for slabs and/or walls

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>ELY</td>
<td>1.</td>
<td>Column's length coefficient to evaluate slenderness effect in local Y axis</td>
</tr>
<tr>
<td>12</td>
<td>ELZ</td>
<td>1.</td>
<td>Column's length coefficient to evaluate slenderness effect in local Z axis</td>
</tr>
</tbody>
</table>
| 13  | RSH            | 1.            | Class of shear reinforcement: Russian Grade:  
- 1 = A240;  
- 2 = A300;  
- 3 = A400;  
- 4 = A500;  
- 5 = B500;  
- 6 = A500SP;  
European grade:  
- 11 = S240;  
- 12 = S400;  
- 13 = S500;  

---

D. Design Codes

STAAD.Pro  
User Manual
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| 2   | RCL            | 3             | Class of longitudinal reinforcement:  
- Russian Grade:  
  - 1 = A240;  
  - 2 = A300;  
  - 3 = A400;  
  - 4 = A500;  
  - 5 = B500;  
  - 6 = A500SP;  
- European Grade:  
  - 11 = S240;  
  - 12 = S400;  
  - 13 = S500; |
<p>| 3   | USM            | 1.0           | Total product of service conditions coefficients for longitudinal reinforcement ($g_s$) |
| 4   | UB2            | 0.9           | Specific service conditions coefficient for concrete ($g_{b2}$) |
| 5   | SDX            | 16.0          | Diameter of reinforcing bars located in the first local (X) direction of slab/wall |
| 6   | SDY            | 16.0          | Diameter of reinforcing bars located in the second local (Y) direction of slab/wall |</p>
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| 7   | BCL            | 15.           | Compression class of concrete:  
• 10 = B10;  
• 15 = B15  
• 20 = B20;  
• 25 = B25;  
• 30 = B30;  
• 35 = B35;  
• 40 = B40;  
• 45 = B45;  
• 50 = B50;  
• 55 = B55;  
• 60 = B60;  
• 8.10 = C8/10  
• 12.15 = C12/15;  
• 16.20 = C16/20  
• 25.30 = C25/30  
• 30.37 = C30/37  
• 35.45 = C35/45  
• 40.50 = C50/50  
• 45.55 = C45/55  
• 50.60 = C50/60  
• 60.75 = C60/75  
• 70.85 = C70/85  
• 80.95 = C80/95  
• 90.105 = C90/105 |
| 8   | UBM            | 1.            | Product of service conditions coefficients for concrete, except UB2 ($B_b$) |
| 9   | TEM            | 0.            | Parameter of concrete hardening conditions:  
• TEM=0, for natural hardening conditions;  
• TEM=1, for steam hardening conditions |
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>CL</td>
<td>0.05</td>
<td>Distance from top/bottom face of slab/wall element to the center of longitudinal reinforcing bars located in first local (X) direction. (Main thickness of top/bottom concrete cover for slab/wall element)</td>
</tr>
<tr>
<td>11</td>
<td>CRA</td>
<td>0.05</td>
<td>Distance from top/bottom face of slab/wall element to the center of transverse reinforcing bars located in second local (Y) direction. (Secondary thickness of top/bottom concrete cover for slab/wall)</td>
</tr>
<tr>
<td>12</td>
<td>WST</td>
<td>0.4</td>
<td>Ultimate width of short-term crack</td>
</tr>
<tr>
<td>13</td>
<td>WLT</td>
<td>0.3</td>
<td>Ultimate width of long-term crack</td>
</tr>
<tr>
<td>No.</td>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-----</td>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| 14  | STA            | 0             | Parameter of limit state for slab/wall design:  
  • STA=0, if calculation of nonsymmetrical reinforcement must be carried out according to the requirements of load carrying capacity (the first limit state);  
  • STA=1, if calculation of symmetrical reinforcement must be carried out according to the requirements of load carrying capacity (the first limit state);  
  • STA=2, if calculation of nonsymmetrical reinforcement must be carried according to the cracking requirements (the second limit state);  
  • STA=3, if calculation of symmetrical reinforcement must be carried according to the cracking requirements (the second limit state) |
<p>| 15  | SELX           | 0.            | Design length of wall member to evaluate slenderness effect in local X axis |
| 16  | SELY           | 0.            | Design length of wall member to evaluate slenderness effect in local Y axis |</p>
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| 17  | MMA            | 0             | Design parameter of slab/wall reinforcement:  
  • MMA=0, if reinforcement calculation must be applied by stresses in local axis;  
  • MMA=1, if reinforcement calculation must be applied by principal stresses |
| 18  | MMB            | 1             | Design parameter of slab/wall reinforcement:  
  • MMB=0, if the effect of additional eccentricity is not taken into account;  
  • MMB=1, if the effect of additional eccentricity is taken into account |
| 19  | RSH            | 1             | Class of shear reinforcement:  
  Russian Grade:  
  • 1 = A240;  
  • 2 = A300;  
  • 3 = A400;  
  • 4 = A500;  
  • 5 = B500;  
  • 6 = A500SP;  
  European grade:  
  • 11 = S240;  
  • 12 = S400;  
  • 13 = S500; |

**D13.A.3 Beams**

Reinforcement for beams of rectangular and T cross-section can be calculated. In calculation of longitudinal reinforcement bending moment about local axis $Z_{loc}$ and torsional moments are considered, but influence of longitudinal forces and bending moments in relation to local axis $Y_{loc}$ is ignored. In calculation of transverse reinforcement shear forces parallel to local axis $Y_{loc}$ and torsional moments are taken into account.
Reinforcement for beams can be calculated either from conditions of strength or from conditions of open crack width limitation (see parameter \( \text{SSE} \)).

Parameters \( \text{SFA} \) and \( \text{EFA} \) are considered only in calculation of transverse reinforcement.

In general case calculation of reinforcement for beams is carried out two times – according to strength conditions and according to conditions of open crack width limitation. In reinforcement calculations from conditions of strength design values of load have to be taken and in calculations from conditions of crack width limitation – characteristic (normative) load values are used. Both calculations can be carried out in one session with the use multiple analysis possibility of the program STAAD.Pro.

In most cases calculation of reinforcement is carried out with account only of a part of loadings. In such cases command LOAD LIST is used, in which numbers of loads considered in calculation are indicated. Number of permanent and long-term loads equal to parameter \( \text{NLT} \) must be included into the list of considered loads.

It has to be noted, that values of parameters \( \text{DD1} \) and \( \text{DD2} \) have influence not only on the width of opened crack but also in some cases, on design and normative reinforcement resistances.

Parameter \( \text{BCL} \) can be equal to any value of concrete compression strength class given in SNiP 2.03.01-84* and to any intermediate value as well.

It should be remembered, that accuracy of results of calculation of transverse reinforcement increases with the value of parameter \( \text{NSE} \).

Parameters \( \text{SFA} \) and \( \text{EFA} \) are considered only in calculations of transverse reinforcement. Beam 1 is shown in Figure 2 with rigid intervals the lengths of which are: at the start of the beam 0.3m and at the end – 0.2m. In modeling of the beam the following command can be used.

```plaintext
MEMBER OFFSET
  1  START 0.3 0 0
  1  END -0.2 0 0
```

When command MEMBER OFFSET is used forces corresponding to the beam the length of which is equal to the distance between points a and b are calculated and then used in calculation of reinforcement. In such case it is necessary to take into account default values of parameters \( \text{SFA} \) and \( \text{EFA} \) equal to zero.

When command MEMBER OFFSET is not used forces corresponding to the beam the length of which is equal to the distance between points 10 and 11 are calculated and then used in calculation of reinforcement. In this case it is necessary to consider values of parameters \( \text{SFA}=0.3 \) and \( \text{EFA}=0.2 \) in reinforcement calculation.

In both cases calculated quantity of transverse reinforcement will be the same. Calculated quantity of longitudinal reinforcement in the second case will be greater.

For beam the following output is generated:

- beam number;
• method of calculation (according to conditions of strength or limitations of opened crack width);
• length and cross-sectional dimensions;
• distance from resultant of forces acting in bottom/top reinforcement to bottom/top edge of the section;
• distance from the side edge of cross-section of the beam web to the centroid of longitudinal bars located at this edge;
• concrete class;
• class of longitudinal and transverse reinforcement;
• assumed in calculations bar diameters of longitudinal and transverse reinforcement;
• calculation results of longitudinal and transverse reinforcement (in two tables).

In nine columns of the first table the following results are presented:

Table 194: Beam design output 1

<table>
<thead>
<tr>
<th>Result</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section</td>
<td>distance of the section from the “start” of the beam, ( \text{mm} )</td>
</tr>
<tr>
<td>As-</td>
<td>cross-sectional area of longitudinal reinforcement in the bottom zone of cross-section of the beam, if angle ( BETA = 0^\circ ), or in the top zone, if ( BETA = 180^\circ ), ( \text{sq.cm} )</td>
</tr>
<tr>
<td>As+</td>
<td>cross-sectional area of longitudinal reinforcement in the top zone of cross-section of the beam, if angle ( BETA = 0^\circ ), or in the top zone, if ( BETA = 180^\circ ), ( \text{sq.cm} )</td>
</tr>
<tr>
<td>Moments (-/+)</td>
<td>values of bending moments, determining cross-sectional areas of longitudinal reinforcement ( As- ) and ( As+ ), ( \text{kNm} )</td>
</tr>
<tr>
<td>Load. N. (-/+)</td>
<td>numbers of loading versions, determining cross-sectional areas of longitudinal reinforcement</td>
</tr>
<tr>
<td>Acrc1</td>
<td>short-term opened crack width*, ( mm )</td>
</tr>
<tr>
<td>Acrc2</td>
<td>long-term opened crack width*, ( mm )</td>
</tr>
</tbody>
</table>

* Opened crack width is presented only in the case when calculation is performed according to conditions limiting opened crack width.

In ten columns of second table the following results are presented:

Table 195: Beam design output 2

<table>
<thead>
<tr>
<th>Result</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section</td>
<td>distance of the section from the “start” of the beam, ( mm )</td>
</tr>
<tr>
<td>Qsw</td>
<td>intensity of transverse reinforcement, ( kN/m )</td>
</tr>
</tbody>
</table>
Asw - cross-sectional area of transverse bars, sq.cm, if their step is 10, 15, 20, 25 or 30 cm

Q - value of shear force parallel to the local axis, kN

T - value of torsional moment, kNm

Load N. - number of loading version, determining intensity of transverse reinforcement

### D13.A.4 Columns

Reinforcement for columns of rectangular or circular cross-section can be calculated. Flexibility of columns can be evaluated in two ways. In the case of usual analysis (command PERFORM ANALYSIS) flexibility is assessed by parameters ELY and ELZ, values of which should conform with recommendation of the Code SNiP 2.03.01-84*. If P-DELTA (analysis according to deformed diagram) or NONLINEAR (nonlinear geometry) analysis is performed, values of parameters ELY and ELZ should be close to zero, for example ELY = ELZ = 0.01.

Longitudinal reinforcement for columns is calculated only from condition of strength. Longitudinal forces and bending moments in relation to local axes $Y_{loc}$ and $Z_{loc}$ are taken into account in longitudinal reinforcement calculations.

For rectangular columns the following output is generated:

- column number;
- column length and cross-sectional dimensions;
- distance of centroid of each longitudinal bar from the nearest edge of the cross-section;
- concrete class;
- longitudinal reinforcement class;
- range of longitudinal reinforcement bar diameters assumed in calculation;
- diameter of longitudinal reinforcement bars obtained in calculation;
- total quantity of longitudinal bars;
- quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis $Y_{loc}$;
- quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis $Z_{loc}$.

In nine columns of the table under the heading LONGITUDINAL REINFORCEMENT the following output is presented:

#### Table 196: Column design output 1

<table>
<thead>
<tr>
<th>Result</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section</td>
<td>distance of the section from the “start” of the column, mm</td>
</tr>
<tr>
<td>Astot</td>
<td>total cross-sectional area of longitudinal reinforcement, sq.cm</td>
</tr>
</tbody>
</table>
Asy: cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis $Y_{loc}$, sq.cm

Asz: cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis $Z_{loc}$, sq.cm

Percent: reinforcement percentage in the section

Nx, Mz, My: respective values of longitudinal force and bending moments in relation to the local axes $Z_{loc}$ and $Y_{loc}$, determining cross-sectional area of longitudinal reinforcement

Load.N.: number of loading version, determining cross-sectional area of longitudinal reinforcement

Diameter of longitudinal reinforcement bars, total quantity of longitudinal bars as well as quantity of longitudinal bars at each edge of the section obtained from calculation should be considered as recommendation. In this case arrangement of reinforcement in the section depends on the orientation of the local axes and is as follows:

Calculated values of reinforcement cross-sectional areas are presented in the table and they may differ from recommended on the lower side.

When it is not possible according to detailing provisions to arrange in the column longitudinal reinforcement determined from calculation additional message is derived.

For columns of circular section the following output is generated:

- column number;
- column length and diameter of cross-section;
- distance of centroid of each longitudinal bar to the edge of cross-section;
- longitudinal reinforcement class;
- assumed in calculation range of diameters of longitudinal reinforcement bars;
- diameter of longitudinal reinforcement bars obtained from calculation;
- quantity of longitudinal bars.

In seven columns of the table under the heading LONGITUDINAL REINFORCEMENT the following results are presented:
Diameter of longitudinal reinforcement bars, total quantity of longitudinal bars as well as quantity of longitudinal bars at each edge of the section should be considered as recommendation.

Arrangement of reinforcement in section in this case is shown below:

Calculated cross-sectional areas of reinforcement presented in the table may differ from recommended on the lower side.

When according to detailing provisions it is not possible to arrange in the column longitudinal reinforcement obtained from calculation additional message is derived.

**D13.A.5 Two Dimensional Elements (slabs, walls, shells)**

In general case calculation of reinforcement for 2D members is carried out two times – according to conditions of strength and conditions of limiting opened width of cracks. If reinforcement is calculated according to conditions of strength, design values of loads have to be used, and for conditions of limiting crack width – characteristic (normative) loads are employed. Both calculations can be made in one session taking advantage of multiple analysis possibility of the program STAAD.Pro.

Symmetric or nonsymmetrical reinforcement of 2D members is calculated according to conditions of strength or according to conditions of limiting opened crack width (see for example STA).

In reinforcement calculation for 2D members it is necessary to pay attention to arrangement of local axes of member and direction of reinforcement (see for example CL and CRA).
An example of output of calculation results is presented bellow.

**SLAB/WALL DESIGN RESULTS**
(by stresses in local axes for limitation of crack width)
### Table 197: Slab design output

<table>
<thead>
<tr>
<th>Element</th>
<th>Asx sq.cm/m</th>
<th>Mx kNm/m</th>
<th>Nx kN/m</th>
<th>Load N. (X)</th>
<th>Asy sq.cm/m</th>
<th>My kNm/m</th>
<th>Ny kN/m</th>
<th>Load N. (Y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>60 TOP</td>
<td>0.00</td>
<td>- 4.9</td>
<td>0.00</td>
<td>1</td>
<td>0.00</td>
<td>- 4.5</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td>BOT</td>
<td>3.53</td>
<td>- 9.9</td>
<td>0.00</td>
<td>3</td>
<td>3.46</td>
<td>- 8.9</td>
<td>0.00</td>
<td>3</td>
</tr>
<tr>
<td>61 TOP</td>
<td>0.00</td>
<td>- 5.3</td>
<td>0.00</td>
<td>1</td>
<td>0.00</td>
<td>- 4.7</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td>BOT</td>
<td>3.87</td>
<td>- 10.7</td>
<td>0.00</td>
<td>3</td>
<td>3.65</td>
<td>- 9.4</td>
<td>0.00</td>
<td>3</td>
</tr>
<tr>
<td>62 TOP</td>
<td>0.00</td>
<td>- 5.6</td>
<td>0.00</td>
<td>1</td>
<td>0.00</td>
<td>- 4.8</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td>BOT</td>
<td>4.10</td>
<td>- 11.2</td>
<td>0.00</td>
<td>3</td>
<td>3.77</td>
<td>- 9.6</td>
<td>0.00</td>
<td>3</td>
</tr>
</tbody>
</table>

Here:

- **Asx sq.cm/m**: intensity of reinforcing in the first direction (parallel to the local axis), sq.cm/m
- **Mx kNm/m**: distributed bending moment in respect to the local axis, kNm/m
- **Nx kN/m**: distributed longitudinal force directed parallel to the axis, kN/m
- **Load N.(X)**: number of loading version, determining intensity of reinforcing in the first direction
- **Asy sq.cm/m**: intensity of reinforcing in the second direction (parallel to the local axis), sq.cm/m
- **My kNm/m**: distributed bending moment in respect to the local axis, kNm/m
- **Ny kN/m**: distributed longitudinal force directed parallel to the local axis, kN/m
- **Load N.(Y)**: number of loading version, determining intensity of reinforcing in the second direction

STAAD.Pro is capable of performing steel design based on the Russian code СНиП II-23-81* Часть II Нормы проектирования Стальные конструкции (SNiP 2.23-81* Part II Design Standards for Steel Construction).

In STAAD.Pro V8i (SELECTseries 5) or later, design of members per SNiP 2.23-81* requires the STAAD ECC. Super Code SELECT Code Pack.

D13.B.1 General

Design Code SNiP Steel Structures –as is the case in the majority of modern codes– is based on the method of limit states. The following groups of limit states are defined in the Code.

- The first group is concerned with losses of general shape and stability, failure, qualitative changes in configuration of structure. Appearance of non-allowable residual deformations, displacements, yielding of materials or opening of cracks.
- The second group is concerned with states of structures making worse normal their service or reducing durability due to not allowable deflections, deviations, settlements, vibrations, etc.

Analysis of structures for the first limit state is performed using the maximum (design) loads and actions, which can cause failure of structures.

Analysis of structures for the second limit state is performed using service (normative) loads and actions. Relation between design and normative loads is referred to as coefficient of load reliability, which is defined in SNiP 2.01.07.- 85 “Loads and Actions”.

Coefficient of reliability for destination GAMA n according to SNiP 2.01.07.- 85 shall be taken in to account determining loads or their combinations.

In this version of the program only members from rolled, tube and roll-formed assortment sections and also from compound such as double angles of T-type sections, double channels are presented. Design of other members of compound section will be presented in other versions of the program.

Economy of selected section is indicated by ratio (RATIO) \( \sigma / R_y \) presented in calculation results. A section is economical when said ratio equals to 0,9 – 0,95.
D13.B.2 Built-in Russian Steel Section Library

Typical sections of members being checked and selected according to SNiP 2.01.07.- 81* are presented in the following tables.

**Table 198: Typical Sections for Russian Steel Design**

<table>
<thead>
<tr>
<th>Section</th>
<th>Section Type</th>
<th>Designation form</th>
</tr>
</thead>
<tbody>
<tr>
<td>I-beam (GOST 8239-89)</td>
<td></td>
<td>ST I12</td>
</tr>
<tr>
<td>Regular I-beam (GOST 26020-83)</td>
<td></td>
<td>ST B1-10</td>
</tr>
<tr>
<td>Broad-flanged I-beam (GOST 26020-83)</td>
<td></td>
<td>ST SH1-23</td>
</tr>
<tr>
<td>Column I-beam (GOST 26020-83)</td>
<td></td>
<td>ST K1-20</td>
</tr>
<tr>
<td>Channel (GOST 8240-89)</td>
<td></td>
<td>ST C14</td>
</tr>
<tr>
<td>Equal legs angle (GOST 8509-89)</td>
<td></td>
<td>ST L100x100x7</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RA L100x100x7</td>
</tr>
<tr>
<td>Unequal legs angle (GOST 8510-89)</td>
<td></td>
<td>ST L125x80x10</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RA L125x80x10</td>
</tr>
<tr>
<td>Pipes (welded and for gas piping)</td>
<td></td>
<td>ST PIP102x5.5</td>
</tr>
<tr>
<td></td>
<td></td>
<td>or</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ST PIPE OD 0.102 ID 0.055</td>
</tr>
</tbody>
</table>
## Section

<table>
<thead>
<tr>
<th>Section Type</th>
<th>Designation form</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roll-formed square and rectangular tubes</td>
<td>ST TUB160x120x3 or ST TUBE TH 0.003 WT 0.12 DT 0.16</td>
</tr>
</tbody>
</table>

*Table 199: Compound Sections for Russian Steel Design*

<table>
<thead>
<tr>
<th>Section</th>
<th>Section Type</th>
<th>Designation form</th>
</tr>
</thead>
<tbody>
<tr>
<td>Double channels</td>
<td></td>
<td>D C14 SP 0.01 (SP – clear distance between channel walls)</td>
</tr>
<tr>
<td>Double equal legs angles</td>
<td></td>
<td>LD L100x100x7 SP 0.01 (SP – clear distance between angle walls)</td>
</tr>
<tr>
<td>Double unequal legs angles with long legs back to back</td>
<td></td>
<td>LD L125x80x10 SP 0.01 (SP – clear distance between angle walls)</td>
</tr>
<tr>
<td>Double unequal legs angles with short legs back to back</td>
<td></td>
<td>SD L125x80x10 SP 0.01 (SP – clear distance between angle walls)</td>
</tr>
<tr>
<td>Tee with flange at the top</td>
<td></td>
<td>T I12 T B1-10 T SH1-23 T K1-20</td>
</tr>
</tbody>
</table>

*Note:* Flange of Tee beam is at the top part of cross-section if beta angle = 0°, or at the bottom part if beta angle = 180°.

For entry of cross-sectional dimensions command `MEMBER PROPERTIES RUSSIAN` is used.

**Example**

```
UNITS METER
MEMBER PROPERTY RUSSIAN
```
D13.B.3 Member Capacities

Algorithms for selection and review of sections for steel members according to assortments and databases of the main rolled steel producers from given countries and according to international standards as well are included in STAAD.Pro program. In this program version only assortment sections can be utilized.

Example

* Command of analysis
PERFORM ANALYSIS
* Command of loadings and their combinations considered in design
LOAD LIST 1 5 TO 9
* Command to start design according to Russian Code
PARAMETER
CODE RUSSIAN
* List of parameters used in checking and selecting
BEAM 1. ALL

Obligatory parameter
LY 4. MEMB 1 TO 4
LZ 4. MEM 1 TO 4
MAIN 1. ALL
SGR 3. ALL
SBLT 0 ALL
* Parameter of output amount of information on calculation results
TRACK 2. ALL
.  
* Command to start section check procedure
CHECK CODE ALL
D13.B.3.1 Axial Tension Members

Stress in a section of axial tension member shall not exceed design strength $R_y$ of selected steel multiplied by coefficient of service conditions $\gamma_c$ (KY and KZ), table 6 of SNiP 2.01.07.-81*. Slenderness of tension member (CMM) shall not exceed slenderness limit indicated in table 20 of SNiP 2.01.07.-81* (default value $\lambda_u = 200$, but another value can be defined). Net section factor (ratio $A_{net}/A_{gross}$ (NSF)) is used for tension member to allow for reduction of design cross-section area.

D13.B.3.2 Axial Compression Members

All axial compression members are calculated as long bars, i.e., with allowance for slenderness ($\lambda = l_0/i_{min}$). The calculation is performed in accordance with the clause 5.3 of SNiP 2.01.07.-81*, buckling coefficient $\phi$ is determined by formula 8-10. Effective bar lengths (within and out of plane) taking into account role and location of the bar in the structure, as well as fixation of ends ($l_0 = \mu l$), are determined according to requirements of chapter 6 or addition 6 to SNiP 2.01.07.-81* and are set by specification of members. Slenderness of compression members (CMN) shall not exceed limit values given in table 19 of SNiP 2.01.07.-81*. Value of coefficient $\alpha$ being used in table 19 is taken within limits from 0.5 to 1.0. Limit slenderness value depends on stress acting in the member, section area, buckling coefficient and design resistance of steel.

Since slenderness can be different in various planes the greatest slenderness is assumed in calculations.

D13.B.3.3 Flexural members

Members subjected to the action of bending moments and shear forces are called flexural members.

Calculation of flexural members consists of verification of strength, stability and deflection.

Normal and tangential stresses are verified by strength calculation of members. Normal stresses are calculated in the outermost section fibres. Tangential stresses are verified in the neutral axis zone of the same section. If normal stresses do not exceed design steel strength and tangential stresses do not exceed design value of steel shear strength $R_s\gamma_s$ then according to clause 5.14 of SNiP 2.01.07.-81* principal stresses are checked.

General stability of member subjected to bending in one plane are calculated in accordance with clause 5.15 of SNiP 2.01.07.-81*, and subjected to bending in two planes – in accordance with "Guide to design of steel structures" (to SNiP 2.01.07.-81*). Coefficient $\phi_b$ value is determined according to appendix 7 of SNiP 2.01.07.-81*. Additional data about load (concentrated or distributed), numbers of bracing restraints of compression flanges, location of applied load are required. For closed sections it is assumed that coefficient $\phi_b = 1.0$.

Simply supported (non-continuous) beams can be calculated in elastic as well as in elastic-plastic state according to requirements of clause 5.18 of SNiP 2.01.07.-81*. Calculation can be selected by specification of structure in input data.
Stiffness of flexural members is verified comparing input value of deflection limit (through parameter DFF) with maximum displacement of a section of flexural member allowing for load reliability coefficient, which is specified, in input data. Limit values of deflection are determined in accordance with SNiP 2.01.07.- 85 “Loads and Actions. Addition chapter 10. Deflections and displacements”. Verification of deflection is performed only in the case of review (CHECK) problem.

**D13.B.3.4 Eccentric Compression/Tension Members**

Eccentric compression or tension members are subjected to simultaneous action of axial force and bending moment. Bending moment appears due to eccentric application of longitudinal force or due to transverse force. Stress in eccentric compression/tension members is obtained as a sum of stresses due to axial force and bending.

Following the requirements of clause 5.25 of SNiP 2.01.07.- 81* resistance of eccentric compression/tension member taking into consideration condition $R_y < 530 \text{ MPa}$, $\tau < 0.5R_y$ and $N/(A_iR_y) > 0.1$ is calculated by formula 49, and in other cases-by formula 50. Calculations of stability verification are performed according to requirements of clauses 5.27, 5.30, 5.32 or 5.34.

Calculation for strength of eccentric tension members is made according to formula 50 of SNiP 2.01.07.- 81*.

When reduced relative eccentricity $m_{ef} > 20$ eccentric compression members are calculated as flexural members ($N = 0$), when $m_{ef} < 20$ strength by formula 49 is not verified (clause 5.24).

**D13.B.4 Design Parameters**

Information on parameters, data used for check and selection of sections in design of steel structures according to Russian Code is presented in the following table.

In this version of calculation according to requirements of SNiP 2.01.07.- 81* there is common database of equal legs angles and unequal legs angles, therefore solution of section selection problem may give equal legs angle as well as unequal legs angle irrespective of set at the beginning. The same is and with rectangular and square tubes.

Values of parameters do not depend on command UNIT. Only these values of parameters, which differ from, defined in the program need to be included in the input data file.

Review of sections (command CHECK) can be performed according to the first and the second group of limit states. Selection of section (command SELECT) can be performed only according to the first group of limit states with subsequent recalculation and verification of selected section with allowance for deflection.

Calculation for the first group of limit states involves selection of members according to strength and stability. Parameters $CMN$ and $CMM$ give opportunity to set slenderness limit for compression and tension members respectively for their stability calculation, or refuse consideration of slenderness by setting default parameters. In this case selection of sections will be performed with consideration only of strength check.

Check for deflection performed by setting parameter DFF (maximum allowable relative deflection value) different from set in the program.

In the case of application of steel not defined by SNiP and/or GOST it is necessary to set their design strength by parameters UNL and PY.

In determination of steel parameters SBLT and MAIN shall be approved (see Table 15B.4).

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.
### Table 200: Parameters for Steel design according to Russian Code (SNiP II – 23 – 81*, edition 1990)

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as RUSSIAN 1990. Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
</tbody>
</table>
| BEAM           | 1             | Member design parameter: 
  - BEAM = 0, Design members for forces at their ends or at the sections defined by SECTION command; 
  - BEAM = 1, Calculate the major axis moment Mz at 13 points along the beam and design beam at the location of maximum Mz; 
  - BEAM = 2, Same as BEAM=1, but additional checks are carried out at beam ends and at critical intermediate section; 
  - BEAM = 3, Calculate forces at 13 points and perform design checks at all locations including the ends |
| CB             | 1             | Place of loading on beam: 
  - CB = 1, for loading on top flange; 
  - CB = 2, for loading on bottom flange |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| CMM            | 0            | Slenderness limit value for tension members:  
|                |              | • CMM = 0, if slenderness is suppressed;  
|                |              | • CMM = 2, if ultimate slenderness value is "150";  
|                |              | • CMM = 2, if ultimate slenderness value is "200";  
|                |              | • CMM = 3, if ultimate slenderness value is "250";  
|                |              | • CMM = 4, if ultimate slenderness value is "300";  
|                |              | • CMM = 5, if ultimate slenderness value is "350";  
|                |              | • CMM = 6, if ultimate slenderness value is "400"  
|                |              | Set slenderness limit value not equal to "0" for design with evaluation of buckling effect |
### Parameter Name | Default Value | Description
--- | --- | ---
CMN | 0 | Slenderness limit value for compression members:
- CMN = 0, if slenderness is suppressed
- CMN = 1, if slenderness limit value is 180-60a
- CMN = 2, if slenderness limit value is 120
- CMN = 3, if slenderness limit value is 210-60a
- CMN = 4, if slenderness limit value is 220-40a
- CMN = 5, if slenderness limit value is 220
- CMN = 6, if slenderness limit value is 180-60a
- CMN = 7, if slenderness limit value is 210-60a
- CMN = 8, if slenderness limit value is 200
- CMN = 9, if slenderness limit value is 150
Set slenderness limit value not equal to "0" for design with evaluation of buckling effect

DFF | 0. | Allowable limit of relative local deflection (Member length/Deflection Ratio):
Default value 0 is valid if design is applied without deflection limitation.
Set for deflection check only

DMAX | 1. | Maximum allowable section depth

DMIN | 0. | Minimum allowable section depth

GAMC1 | 1.0 | Specific service condition coefficient for buckling design
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GAMC2</td>
<td>1.0</td>
<td>Specific service condition coefficient for strength design</td>
</tr>
<tr>
<td>GMF</td>
<td>1.0</td>
<td>Ratio between design and characteristic load value</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Coefficient of effective length in respect to local axis Y (in plane XZ)</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Coefficient of effective length in respect to local axis Z (in plane XY)</td>
</tr>
<tr>
<td>LEG</td>
<td>4</td>
<td>Type and position of loading on beam:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• LEG = 1, for loading concentrated in the middle span;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• LEG = 2, for loading concentrated in the quarter of the span;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• LEG = 3, for loading concentrated at the end of bracket;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• LEG = 4, for loading uniformly distributed on beam;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• LEG = 5, for loading uniformly distributed on bracket</td>
</tr>
<tr>
<td>LY [m]</td>
<td>Member length</td>
<td>Effective length in respect to local axis Y (in plane XZ)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Default is selected member’s length</td>
</tr>
<tr>
<td>LZ [m]</td>
<td>Member length</td>
<td>Effective length in respect to local axis Z (in plane XY)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Default is selected member’s length</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| MAIN           | 1            | Standard of steel grade (GOST):  
- MAIN = 1, if Standard of steel grade is GOST27772-88;  
- MAIN = 2, if Standard of steel grade is GOST10705-80;  
- MAIN = 3, if Standard of steel grade is GOST10706-76;  
- MAIN = 4, if Standard of steel grade is GOST8731-87;  
- MAIN = 5, if Standard of steel grade is TY14-3-567-76 |
| NSF            | 1.0          | Net section factor for tension members or web section area weakening factor for bending members |
| PY [MPa]       | 0            | Design steel strength (yield strength):  
If parameters MAIN according to Standard of steel grade (GOST) and by SGR according to Steel grade (STAL) are not defined |
| RATIO          | 1.0          | Permissible ratio of the actual capacities. |
| SBLT           | 0            | Number of lateral bracing restraints along the span:  
- SBLT = 0, if beam not fixed;  
- SBLT = 1, one restraint in the middle of the span;  
- SBLT = 2, 3, etc. number of uniformly spaced lateral supports along the span |
<p>| SGR            | 1            | Steel grade (STAL). Refer to Table 12B.4 below. |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| TB             | 0             | Indication of elastic or elastic-plastic calculation:  
                     • TB = 0, for elastic calculation  
                     • TB = 1, for elastic-plastic calculation  
                     Set for members under bending or non-axial compression/tension only. |
| TRACK          | 0             | Output parameter:  
                     • TRACK = 0, for suppressed output information;  
                     • TRACK = 1, for extended output information;  
                     • TRACK = 2, for advanced output information |
| UNL [MPa]      | 0             | Design steel strength (ultimate strength):  
                     If parameters MAIN according to Standard of steel grade (GOST) and by SGR according to Steel grade (STAL) are not defined |

Table 201: Steel types for design of steel structures according to SNiP 2.01.07.-81* (table 51 and 51a)

<table>
<thead>
<tr>
<th>SGR Value</th>
<th>Steel</th>
<th>Parameter MAIN</th>
<th>GOST</th>
<th>For members*</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>C235</td>
<td>1</td>
<td>GOST 27772-88</td>
<td>GT, F</td>
</tr>
<tr>
<td>2</td>
<td>C245</td>
<td>1</td>
<td>“</td>
<td>GT, F</td>
</tr>
<tr>
<td>3</td>
<td>C255</td>
<td>1</td>
<td>“</td>
<td>GT, F</td>
</tr>
<tr>
<td>4</td>
<td>C275</td>
<td>1</td>
<td>“</td>
<td>GT, F</td>
</tr>
<tr>
<td>5</td>
<td>C285</td>
<td>1</td>
<td>“</td>
<td>GT, F</td>
</tr>
<tr>
<td>6</td>
<td>C345</td>
<td>1</td>
<td>“</td>
<td>GT, F</td>
</tr>
<tr>
<td>7</td>
<td>C345K</td>
<td>1</td>
<td>“</td>
<td>GT, F</td>
</tr>
<tr>
<td>8</td>
<td>C375</td>
<td>1</td>
<td>“</td>
<td>GT, F</td>
</tr>
<tr>
<td>9</td>
<td>C390</td>
<td>1</td>
<td>“</td>
<td>F</td>
</tr>
</tbody>
</table>
### D13.B.5 Member Selection and Code Check

Both code checking and member selection options are available in SNiP 2.23-81*.

Output of selection and check results are given in suppressed, extended and advanced forms. Form of output results depends on value of parameter TRACK.

Results are presented in tables. Three versions of output results are possible: suppressed – results according the critical strength condition (TRACK=0), extended - results according to all check conditions (TRACK=1) and advanced – complete information on results of member design (TRACK=2).

In tables of results common data for all TRACKs are indicated:

\[
\text{(TRACK}=2)\]

In tables of results common data for all TRACKs are indicated:

- number of member;
- type and number of cross-section;
- result obtained (ACCEPTED – requirements are met, FAILURE – are not met);
- abbreviated name of normative document (code, standard) (SNiP);
- number of check clause;
- safety of strength (ratio between design and normative values);
- number of the most unfavorable loading;

---

<table>
<thead>
<tr>
<th>SGR Value</th>
<th>Steel</th>
<th>Parameter MAIN</th>
<th>GOST</th>
<th>For members*</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>C390K</td>
<td>1</td>
<td>&quot;</td>
<td>F</td>
</tr>
<tr>
<td>11</td>
<td>C440</td>
<td>1</td>
<td>&quot;</td>
<td>F</td>
</tr>
<tr>
<td>12</td>
<td>C590</td>
<td>1</td>
<td>&quot;</td>
<td>F</td>
</tr>
<tr>
<td>13</td>
<td>C590K</td>
<td>1</td>
<td>&quot;</td>
<td>F</td>
</tr>
<tr>
<td>14</td>
<td>BSt3kp</td>
<td>2</td>
<td>GOST 10705-80*</td>
<td>Tube</td>
</tr>
<tr>
<td>15</td>
<td>BSt3ps</td>
<td>2</td>
<td>GOST 10705-80*</td>
<td>Tube</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3</td>
<td>GOST 10706-76*</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>BSt3sp</td>
<td>2</td>
<td>GOST 10705-80*</td>
<td>Tube</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3</td>
<td>GOST 10706-76*</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>20</td>
<td>4</td>
<td>GOST 8731-87</td>
<td>Tube</td>
</tr>
<tr>
<td>18</td>
<td>16G2AF</td>
<td>5</td>
<td>TY 14-3-567-76</td>
<td>Tube</td>
</tr>
</tbody>
</table>

*GT – members from sheet and roll-formed tubes
F – rolled section steel
value of longitudinal force acting in the member with subscript indicating its direction ("C" – compression, "P" – tension); 
bending moments in relation to local member axes Z and Y; 
distance to section, in which the most unfavorable combination of forces acts.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

Note: The output for the Russian code uses the axes system from STAAD.Pro, except where details of the equations from the code are printed, in which case the axes system is as defined in the code.

**D13.B.5.1 Example of TRACK 0 output**

In suppressed form (TRACK 0) results are presented according to the critical check for given member with indication of SNiP clause number, according to which strength safety of the member is minimum.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>CROSS SECTION NO.</th>
<th>RESULT/ RX</th>
<th>CRITICAL COND/ MZ</th>
<th>RATIO/ MY</th>
<th>LOADING/ LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>I60</td>
<td>PASS</td>
<td>SNiP- 5.18</td>
<td>0.68</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00E+00</td>
<td>-4.650E+02</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

**D13.B.5.2 Example of TRACK 1 output**

In extended form (TRACK 1) results are presented on the basis of all required by SNiP checks for given stress state.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>CROSS SECTION NO.</th>
<th>RESULT/ RX</th>
<th>CRITICAL COND/ MZ</th>
<th>RATIO/ MY</th>
<th>LOADING/ LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>I60</td>
<td>PASS</td>
<td>SNiP- 5.18</td>
<td>0.68</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00E+00</td>
<td>-4.650E+02</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>1</td>
<td>I60</td>
<td>PASS</td>
<td>SNiP- DISPL</td>
<td>0.36</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00E+00</td>
<td>-4.650E+02</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

**D13.B.5.3 Example of a TRACK 2 output**

In advanced form (TRACK=2) in addition to tabled results supplementary information is presented.

Material characteristics:
Steel;
Design resistance;
Elasticity modulus;
Section characteristics:
Length of member;
Section area;
Net area;
Inertia moment (second moment of area) (I);
Section modulus (W);
First moment of area (S);
Radius of gyration;
Effective length;
Slenderness;
Results are presented in two columns, Z and Y respectively.

Design forces:
Longitudinal force;
Moments;
Shear force.

Signs “+” and “-” indicate direction of acting longitudinal force, bending moments and shear forces in accordance with sign rules assumed in program STAAD.

Check results in advanced form are presented with values of intermediate parameters by formulas in analytical and numerical expression with indication of SNiP clause.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>CROSS</th>
<th>RESULT/ CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SECTION NO.</td>
<td>FX</td>
<td>MZ</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>I60</td>
<td>PASS</td>
<td>SNiP- 5.18</td>
</tr>
<tr>
<td></td>
<td>0.000E+00</td>
<td>-4.650E+02</td>
<td>0.000E+00</td>
<td>3.000E+00</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>I60</td>
<td>PASS</td>
<td>SNiP- DISPL</td>
</tr>
<tr>
<td></td>
<td>0.000E+00</td>
<td>-4.650E+02</td>
<td>0.000E+00</td>
<td>3.000E+00</td>
</tr>
</tbody>
</table>

MATERIAL DATA
Steel = C245
Modulus of elasticity = 206.0E+06 KPA
Design Strength (Ry) = 240.0E+03 KPA

SECTION PROPERTIES (units - m)
Member Length = 6.00E+00
Gross Area = 1.38E-02
Net Area = 1.38E-02
Moments (I): 768.0E-06, 173.0E-07
z-axis, y-axis
First moment of area (S): 149.0E-05, 156.0E-06
z-axis, y-axis
Radius of gyration (i): 236.0E-03, 354.0E-04
Effective Length = 600.0E-02, 600.0E-02
Slenderness = 0.00E+00, 0.00E+00

DESIGN DATA (units -kN,m)SNiP II-23-81*/1998
Axial force = 0.00E+00
Moments = -465.0E+00, 0.00E+00
Shear force = 0.00E+00, 500.0E-02

CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
F.(39) M/(C1*Wmin)=-465.0E+00/ 1.12E+00* 2.56E-03= 162.1E+03
F.(41) Q/(H*T)= 500.0E-02/ 6.00E-01* 1.20E-02= 694.E+00
RY*GAMAC= 240.0E+03
ACTUAL SECTION DISPLACEMENT = 1.094E-02 M
MAXIMUM MEMBER DEFLECTION = 1.094E-02 M Loading No. 1
ULTIMATE ALLOWABLE DEFLECTION VALUE = 3.000E-02 M
Conventional notations assumed in presentation of results: "+', '-', '/', '*', **", "SQRT", their respective meanings (i.e., addition, subtraction, division, multiplication, raising to the second power (squared), and square root). Conventional notations of stresses, coefficients and characteristics of steel resistance comply with accepted in the SNiP standard. Only Greek letters are changed by their names (e.g., γ - GAMAC; α - ALPHA; β - BETA, η - ETA, φ - PHI, etc.).

D13.C. Russian Codes - Steel Design Per SP 16.13330.2011

STAAD.Pro is capable of performing steel design based on the Russian code Сн 16.13330.2011 стальные конструкции (SP 16.13330.2011 Steel Structures).

**Note:** This code supersedes SNiP II-23-81*.


**Note:** This feature requires STAAD.Pro 2007.10.10.xx or greater.

D13.C.1 General

Design Code SP Steel Structures –as is the case in the majority of modern codes– is based on the method of limit states. The following groups of limit states are defined in the Code.

- The first group concerns losses of general shape and stability, failure, and qualitative changes in configuration of the structure (i.e., ultimate limit states). Appearance of non-allowable residual deformations, displacements, yielding of materials or opening of cracks.

  Analysis of structures for the first limit state is performed using the maximum (design) loads and actions, which can cause failure of structures.

- The second group concerns states of the structure which worsen their service or reduce durability due to exceeding allowable deflections, deviations, settlements, vibrations, etc. (i.e., service conditions)

  Analysis of structures for the second limit state is performed using service (normative) loads and actions. Relation between design and normative loads is referred to as coefficient of load reliability, which is defined in SNiP 2.01.07.- 85 “Loads and Actions”.

The coefficient of reliability for destination GAMA n according to SP 20.13330.2011 shall be taken in to account determining loads or their combinations.

According to the European standards, the strength of steel is represented by the characteristic value. To obtain the design value, the steel reliability coefficient Gamm is used. The default value of Gamm is 1.0.

**Note:** If there are doubts about the conformance of the European steel type and the standard, it is necessary that the steel type designation used by the ENSGR parameter be set the same as the standard selected in the ENMAIN parameter. If the chosen steel type is not present in the chosen standard, then the program exits with the error code MEMBER NO.145 STEEL S275 IS NOT PRESENT IN EN 10025-6 * ERROR * The calculation will be terminated if the thickness of the designed member web or flange thickness is outside the limits of the steel standard. The error code will be issued: MEMBER NO. 1000 CURRENT THICKNESS IS OUT OF EN 100219-1 * ERROR *

Only members from rolled, tube, and roll-formed assortment sections, and compound sections (such as double angles of T-type sections, double channels) are may be designed in STAAD.Pro.

Economy of selected section is indicated by ratio the ration (which can be set using the RATIO parameter) \( \sigma / R_y \) presented in calculation results. A section is economical when said ratio equals to 0.9 – 0.95.
D13.C.2 Member Capacities

D13.C.2.1 Axial Tension Members

Stress in a section of an axial tension member shall not exceed design strength $R_y$ of the selected steel multiplied by the coefficient of service conditions, $\gamma_c$ (input by the GAM1 and CAMC2 parameters), take from table 1 of SP 16.13330.2011.

The slenderness of tension members shall not exceed the slenderness limit, $\lambda_u$ indicated in table 33 of SP 16.13330.2011 as equal to 150. This limit may be specified using the CMM parameter, which defaults to 150.

The net section factor (the ratio of $A_{net}/A_{gross}$) is specified by the NSF parameter and is used for tension members to allow for the reduction of design cross-section area.

D13.C.2.2 Axial Compression Members

All axial compression members are calculated as long bars (i.e., with allowance for slenderness - $\lambda = l_0/i_{min}$, where $l_0$ is the effective length of the element). Calculation is performed in accordance with clause 7.1.1 of SP 16.13330.2011, with the buckling coefficient $\phi$ determined by equation 8. Effective lengths of elements (within and out-of plane) take into account role and location of the bar in the structure, as well as fixity of the ends ($l_0 = \mu_1$), are determined according the requirements of section 10.3 of SP 16.13330.2011 and are set by the user specification of the members. Slenderness parameters, $\mu_{xz}$ and $\mu_y$, are set using the parameters KZ and KY, respectively. If the slenderness parameters of an element is not precisely known, then the effective length can be specified using the LY and LZ parameters, instead. The ultimate slenderness of compression members shall now exceed the limit values given in table 32 of SP 16.13330.2011, or a user-specified value provided through the CMN parameter. The value of the coefficient $\alpha$ used in Table 32 is taken within the limits of 0.5 and 1.0. The limiting slenderness value in compression elements depends on stress acting the member, buckling coefficient, and design resistance of the steel.

Since the slenderness can be different in various planes, the greatest slenderness ratio is assumed in calculations. A warning is given if the slenderness ratio of a compression element exceeds the limit, but the calculations are continued. If the slenderness ratio exceeds the limit value, the output line containing the slenderness check is preceded by a # (pound or hash symbol).

The calculations of single members are performed in this manner. If the member is subjected to axial forces and bending moment (e.g., due to self weight), then the calculation of load bearing capacity will be done taking into account the axial force and bending moments and the buckling resistance only under the axial compression according to clause 7.1 of SP 13330.2011. Local buckling of the web and flanges of centrally loaded members is checked. Stiffener ribs are recommended if necessary.

D13.C.2.3 Flexural Members

Member subjected to bending moments and shear forces are called flexural members.

There are three classes of flexural elements:

1. Elastic - in cross-section, the stress in the extreme compression fiber of the steel member –assuming an elastic distribution of stresses– can reach the yield strength. $\sigma \leq R_y$, where $\sigma$ is the absolute value of the stress.

2. Elasto-plastic - in part of the cross-section, the stresses are $\sigma \leq R_y$ and in another $\sigma = R_y$.

3. Plastic state (i.e., conditional plastic hinge) - across the entire cross-section, the stresses are $\sigma = R_y$.

The parameter TB is used to specify either class 1 (elastic) or class 2 (elasto-plastic).

The calculation of flexural members consists of verification of strength, stability, and deflection.
Normal and tangential stresses are verified by strength calculations of members. Normal stresses are calculated in the outermost section fibers. Tangential stresses are verified in the neutral axis zone of the same section. If the normal stresses do not exceed design steel strength and tangential stresses do not exceed the design value of steel shear strength, \( R_s \gamma_s \), then according to clause 8.2.1 of SP 16.13330.2011 the principal stresses are checked. For elements subjected to biaxial bending moments according to clauses 8.2.1 and 8.4.1 of SP 16.13330.2011

**D13.C.2.4 Eccentrically Compressed/Tensioned Members**

Eccentrically compressed or tensioned members are subjected to simultaneous action of axial force and bending moment. Bending moment appears to eccentrically applied longitudinal force or due to transverse force.

**D13.C.3 Built-in Russian Steel Section Library**

STAAD.Pro can check many standard sections in SP 16.13330.2011.

**Table 202: Typical Sections for Russian Steel Design**

<table>
<thead>
<tr>
<th>Section</th>
<th>Section Type</th>
<th>Designation form</th>
</tr>
</thead>
<tbody>
<tr>
<td>I-beam (GOST 8239-89)</td>
<td></td>
<td>ST I12</td>
</tr>
<tr>
<td>Regular I-beam (GOST 26020-83)</td>
<td></td>
<td>ST B1-10</td>
</tr>
<tr>
<td>Broad-flanged I-beam (GOST 26020-83)</td>
<td></td>
<td>ST SH1-23</td>
</tr>
<tr>
<td>Column I-beam (GOST 26020-83)</td>
<td></td>
<td>ST K1-20</td>
</tr>
<tr>
<td>Channel (GOST 8240-89)</td>
<td></td>
<td>ST C14</td>
</tr>
<tr>
<td>Equal legs angle (GOST 8509-89)</td>
<td></td>
<td>ST L100x100x7</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RA L100x100x7</td>
</tr>
<tr>
<td>Unequal legs angle (GOST 8510-89)</td>
<td></td>
<td>ST L125x80x10</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RA L125x80x10</td>
</tr>
</tbody>
</table>
### Section Type

<table>
<thead>
<tr>
<th>Section Type</th>
<th>Designation form</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pipes (welded and for gas piping)</td>
<td>ST PIP102X5.5 or ST PIPE OD 0.102 ID 0.055</td>
</tr>
<tr>
<td>Roll-formed square and rectangular tubes</td>
<td>ST TUB160X120X3 or ST TUBE TH 0.003 WT 0.12 DT 0.16</td>
</tr>
</tbody>
</table>

### Table 203: Compound Sections for Russian Steel Design

<table>
<thead>
<tr>
<th>Section Type</th>
<th>Designation form</th>
</tr>
</thead>
<tbody>
<tr>
<td>Double channels</td>
<td>D C14 SP 0.01 (SP – clear distance between channel walls)</td>
</tr>
<tr>
<td>Double equal legs angles</td>
<td>LD L100X100X7 SP 0.01 (SP – clear distance between angle walls)</td>
</tr>
<tr>
<td>Double unequal legs angles with long legs back to back</td>
<td>LD L125X80X10 SP 0.01 (SP – clear distance between angle walls)</td>
</tr>
<tr>
<td>Double unequal legs angles with short legs back to back</td>
<td>SD L125X80X10 SP 0.01 (SP – clear distance between angle walls)</td>
</tr>
<tr>
<td>Tee with flange at the top</td>
<td>T I12 T B1-10 T SH1-23 T K1-20</td>
</tr>
</tbody>
</table>

**Note:** Flange of Tee beam is at the top part of cross-section if beta angle = 0°, or at the bottom part if beta angle = 180°.
For entry of cross-sectional dimensions command MEMBER PROPERTIES RUSSIAN is used. Refer to D13.B.2 Built-in Russian Steel Section Library (on page 2138) for an example.

**D13.C.4 Design Parameters**

**Table 204: Design parameters for design of steel members per SP 16.13330.2011**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as RUSSIAN Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BEAM</td>
<td>1</td>
<td>Member design parameter: 0. Design member for forces at their ends or at the sections defined by the SECTION command (TR.41 Section Specification (on page 2839)). 1. Calculate the major axis moment, Mz, and 13 points along the beam and design the beam the location of the maximum Mz value. 2. Same as BEAM = 1, but additional checks are carried out at beam ends and at critical intermediate sections. 3. Calculate forces at 13 points and perform design checks at all locations, including at ends</td>
</tr>
<tr>
<td>CB</td>
<td>1</td>
<td>Location of loading on the beam: 1. top flange 2. bottom flange</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| CMM            | 0             | Slenderness limit for tension members:  
|                |               | 0. suppress slenderness check  
|                |               | 1. 150  
|                |               | 2. 200  
|                |               | 3. 250  
|                |               | 4. 300  
|                |               | 5. 350  
|                |               | 6. 400  |
| CMN            | 0             | Ultimate slenderness for compression members. Limits as per SP 16.13330.2011 clause 10.4, Table 32:  
|                |               | 0. suppress slenderness check  
|                |               | 1. 180-60a  
|                |               | 2. 120  
|                |               | 3. 210-60a  
|                |               | 4.220-40a  
|                |               | 5. 220  
|                |               | 6. 180-60a  
|                |               | 7. 210-60a  
|                |               | 8. 200  
|                |               | 9. 150  
<p>|                |               | where a = alpha calculated as defined in SP 16.13330.2011 Table 32. |
| DFF            | None          | &quot;Deflection Length&quot; / Maximum allowable local deflection |
| DJ1            | Start Joint of member |
| DJ2            | End Joint of member |
| DMAX           | 1,000 in      | Maximum allowable section depth |
| DMIN           | 0             | Minimum allowable section depth |
| ENMAIN         | 1             | The number of the steel standard taken from Table 3.1 in EN 1993-1-1: 2005 |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENSGR</td>
<td>1</td>
<td>The number of the steel grade from Table 3.1 in EN 1993-1-1: 2005. Also, see Note 1 below.</td>
</tr>
<tr>
<td>GAMC1</td>
<td>1.0</td>
<td>Specific service condition coefficient for buckling design</td>
</tr>
<tr>
<td>GAMC2</td>
<td>1.0</td>
<td>Specific service condition coefficient for strength design</td>
</tr>
<tr>
<td>GAMM</td>
<td>1.0</td>
<td>Specific partial coefficient for steel for European. Refer to D13.C.1 General for details.</td>
</tr>
<tr>
<td>GMF</td>
<td>1.0</td>
<td>Ratio between design and characteristic load value</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>Coefficient of effective length in respect to local axis Y (in plane XZ)</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>Coefficient of effective length in respect to local axis Z (in plane XY)</td>
</tr>
<tr>
<td>LEG</td>
<td>4</td>
<td>Describes the type of and position of loading on the beam:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. concentrated in middle of the span</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. concentrated in the quarter of the span</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. concentrated at the end of bracket</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4. uniformly distributed along beam</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5. uniformly distributed on bracket</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Effective length in respect to local axis Y (in plane XZ)</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Effective length in respect to local axis Z (in plane XY)</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| MAIN           | 1             | Standard of steel grade (GOST):  
|                |               | 1. GOST27772-88  
|                |               | 2. GOST10705-80  
|                |               | 3. GOST10706-76  
|                |               | 4. GOST8731-87  
|                |               | 5. TY14-3-567-76  
| NSF            | 1.0           | Net section factor for tension members or web section area weakening factor for bending members  
| PY             | 0             | Design steel strength (yield strength)  
|                |               | **Note:** Used when parameters MAIN and SGR are not defined for Russian steel materials or ENMAIN and ENSGR for European steel materials. See Note 1 below.  
| RATIO          | 1.0           | Permissible ratio of the actual capacities.  
| SBLT           | 0             | Number of lateral bracing restraints along the span:  
|                |               | 0. beam is not fixed laterally  
|                |               | 1. single restraint at mid-span  
<p>|                |               | 2. or higher is the number of uniformly spaced lateral restraints along the span |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGR</td>
<td>1</td>
<td>Steel grade (STAL): See Note 1 below.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. S235</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. S245</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. S255</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4. S285</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5. S345</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6. S345K</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7. S375</td>
</tr>
<tr>
<td></td>
<td></td>
<td>8. S390</td>
</tr>
<tr>
<td></td>
<td></td>
<td>9. S440</td>
</tr>
<tr>
<td></td>
<td></td>
<td>10. S590</td>
</tr>
<tr>
<td></td>
<td></td>
<td>11. S590K</td>
</tr>
<tr>
<td></td>
<td></td>
<td>12. VSt3kp</td>
</tr>
<tr>
<td></td>
<td></td>
<td>13. VSt3ps</td>
</tr>
<tr>
<td></td>
<td></td>
<td>14. VSt3sp</td>
</tr>
<tr>
<td></td>
<td></td>
<td>15. VSt3sp4</td>
</tr>
<tr>
<td></td>
<td></td>
<td>16. St3sp4</td>
</tr>
<tr>
<td></td>
<td></td>
<td>17. 20</td>
</tr>
<tr>
<td>TB</td>
<td>1</td>
<td>Indication of elastic or elastic-plastic calculation for bending members (non-axially compressed or tensioned) per Cl. 8.1:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. 1st Class (Elastic)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. 2nd Class (Elasto-plastic)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. 3rd Class (Plastic)</td>
</tr>
<tr>
<td>TRACK</td>
<td>0</td>
<td>Output details:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. suppressed output information</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. extended output information for the critical section under the critical load case</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. full output information for the critical section under the critical load case</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. extended output information for all sections under all load cases.</td>
</tr>
</tbody>
</table>
### Parameter Name

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNL</td>
<td>0</td>
<td>Design steel strength (ultimate strength)</td>
</tr>
</tbody>
</table>

**Note:** Used when parameters MAIN and SGR are not defined.

### Notes

1. It is common practice to design only Russian or European shapes to the SNiP codes. For Russian steel sections, the steel design strength value, $R_y$, of a particular steel grade can be obtained from Table C.5 of SP 16.13330.2011 and is used for the SGR parameter. For European sections, the ENSGR and ENMAIN parameters are used accordingly. If steel sections from other countries must be used, the PY parameter is used to specify the steel strength.

### D13.C.5 Member Selection and Code Check

Both code checking and member selection options are available in SP 16.13330.2011.

Output of selection and check results are given in suppressed, extended and advanced forms. Form of output results depends on value of parameter TRACK.

Results are presented in tables. Three versions of output results are possible: suppressed – results according the critical strength condition (TRACK=0), extended - results according to all check conditions (TRACK=1) and advanced – complete information on results of member design (TRACK=2).

In tables of results common data for all TRACKs are indicated:

- (TRACK=2)

In tables of results common data for all TRACKs are indicated:

- number of member;
- type and number of cross-section;
- result obtained (ACCEPTED – requirements are met, FAILURE – are not met);
- abbreviated name of normative document (code, standard) (SNiP);
- number of check clause;
- safety of strength (ratio between design and normative values);
- number of the most unfavorable loading;
- value of longitudinal force acting in the member with subscript indicating its direction (“C” – compression, “P” – tension);
- bending moments in relation to local member axes Z and Y;
- distance to section, in which the most unfavorable combination of forces acts.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.
**Note:** The output for the Russian code uses the axes system from STAAD.Pro, except where details of the equations from the code are printed, in which case the axes system is as defined in the code.

### D13.C.5.1 Example of TRACK 0 output

In suppressed form (TRACK 0) results are presented according to the critical check for given member with indication of SP 16 clause number, according to which strength safety of the member is minimum.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>CROSS SECTION NO.</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.4.1</td>
<td>0.76</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

### D13.C.5.2 Example of TRACK 1 output

In extended form (TRACK 1) results are presented on the basis of all required by SP 16 checks for given stress state.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>CROSS SECTION NO.</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.2.1</td>
<td>0.22</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>2</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.2.1</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>2</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.2.1</td>
<td>0.20</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>2</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.4.1</td>
<td>0.76</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

### D13.C.5.3 Example of a TRACK 2 output

In advanced form (TRACK=2) in addition to tabled results supplementary information is presented.

- Material characteristics:
  - Steel;
  - Design resistance;
  - Elasticity modulus;
- Section characteristics:
  - Length of member;
  - Section area;
  - Net area;
  - Inertia moment (second moment of area) (I);
  - Section modulus (W);
  - First moment of area (S);
  - Radius of gyration;
  - Effective length;
  - Slenderness;
Results are presented in two columns, Z and Y respectively.

Design forces:
- Longitudinal force;
- Moments;
- Shear force.

Signs “+” and “-” indicate direction of acting longitudinal force, bending moments and shear forces in accordance with sign rules assumed in program STAAD.

Check results in advanced form are presented with values of intermediate parameters by formulas in analytical and numerical expression with indication of SNiP clause.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>CROSS SECTION NO.</th>
<th>RESULT/ FX</th>
<th>CRITICAL COND/ MZ</th>
<th>RATIO/ MY</th>
<th>LOADING/ LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 I</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.2.1</td>
<td>0.22</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
<td>4.000E+00</td>
</tr>
<tr>
<td>2 I</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.2.1</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
<td>4.000E+00</td>
</tr>
<tr>
<td>2 I</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.2.1</td>
<td>0.20</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
<td>4.000E+00</td>
</tr>
<tr>
<td>2 I</td>
<td>IPE300</td>
<td>PASS</td>
<td>SNiP- 8.4.1</td>
<td>0.76</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.000E+00</td>
<td>-4.446E+01</td>
<td>0.000E+00</td>
<td>4.000E+00</td>
</tr>
</tbody>
</table>

MATERIAL DATA
- Steel = S355 EN10025-2
- Modulus of elasticity = 206.0E+03 kPa
- Design Strength (Ry) = 355.0E+03 kPa

SECTION PROPERTIES (units - m)
- Member Length = 8.00E+00
- Gross Area = 5.38E-03
- Net Area = 5.38E-03
  - z-axis: 836.0E-07
  - y-axis: 604.0E-08
- Section modulus (W): 557.0E-06
- First moment of area (S): 314.0E-06
- Radius of gyration (i): 125.0E-03
- Effective Length: 160.0E-01
- Slenderness: 0.00E+00

DESIGN DATA (units -kN,m)SNiP II-23-81*/2011
- Axial force: 0.00E+00
  - z-axis: 0.00E+00
  - y-axis: 0.00E+00
- Moments: -445.0E-01
  - z-axis: 0.00E+00
  - y-axis: 0.00E+00
- Shear force: 0.00E+00
  - z-axis: 0.00E+00
  - y-axis: 0.00E+00
- Bi-moment: 0.00E+00

CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
- F.(41) $M/(Wn,\min*100*Ry*GAMAc)=-444.6E-01/( 5.57E-04* 355.0E+03* 1.00E+00= 2.25E-01<1$
- F.(44) $0.87/(Ry*GAMAc)*SQRT(SIGM^2+3*TAUzy^2)=0.87/( 355.0E+03* 1.00E+00)*SQRT(-798.2E+02^2+3* 0.000E+00)= 1.96E-01<1$
- TAUzy/(Rs*GAMAc)= 0.000E+00/( 205.9E+03* 1.00E+00)= 0.00E+00<1
Conventional notations assumed in presentation of results: "+", "-", "/", "*", "**", "SQRT", their respective meanings (i.e., addition, subtraction, division, multiplication, raising to the second power (squared), and square root). Conventional notations of stresses, coefficients and characteristics of steel resistance comply with accepted in the SP standard. Only Greek letters are changed by their names (e.g., \( \gamma_c \)-GAMAC; \( \alpha \)-ALPHA; \( \beta \)-BETA, \( \eta \)-ETA, \( \phi \)-PHI, etc.).

**D13.D. Russian Codes - Concrete Design Per SP 63.1330.2012**

STAAD.Pro is capable of performing concrete design for beams, columns, slabs, and walls based on the Russian code Сп63.1330.2012 Бетонные и железобетонные конструкции. Основные положения (SP 63.1330.2012 Concrete and Reinforced Concrete Structures. Basic Provisions).

**Note:** This code supersedes SNiP 2.03.01-84*.

Design of members per SP 63.1330.2012 requires the **STAAD ECC Super Code SELECT Code Pack**.

**Note:** This feature requires STAAD.Pro 2007.11 or greater.

**D13.D.1 General**

Russian Code SP 63.1330.20121 Concrete and Reinforced Concrete Structures - Basic Provisions is based on the method of limit states. This code defines two groups of limit states.

Analysis according to the first group of limit states is performed to avoid the following phenomena:

- brittle, plastic or other type of failure,
- loss by structure of stable form or position,

Analysis according to the second group of limit states is performed to avoid the following phenomena:

- excessive opening of cracks (if they are allowed according to service conditions)

Analysis of structures for the first group of limit states is performed for the maximum (design) loads and actions. Analysis of structures for the second group of limit states is made in accordance with the operational (normative) loads and actions. Ratio between design and normative loads is called reliability coefficient for loads which is determined according to SNiP 2.01.07.-85 Loads and actions.

STAAD.Pro calculates reinforcement for concrete members. Algorithms for calculation of reinforcement of concrete linear (beams, columns) and 2D (two dimensional) (slabs, walls, shells) members are incorporated into STAAD.Pro. In addition to SP5 2.13330.2011, the Guide for design of plain concrete and reinforced concrete structures from normal weight and lightweight concrete (to SNiP 2.03.01-84) has been incorporated in the calculation algorithms.

STAAD.Pro calculates reinforcement for beams of rectangular or T section and for columns of rectangular or circular section (Fig.1).
The flange of T-shape beams may be situated at the top zone of the section if the angle BETA=0°, or at the bottom zone of the section, if BETA=180°.

**D13.D.2 Design Parameters**

Values of parameters do not depend on the `UNIT` command.

Commands for calculation of reinforcement are located in the input data file after the command of analysis and as a rule, after output commands to print results of calculation.

**Table 205: Parameters for concrete design according to Russian Code SP 63.1330.2012**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be assigned as RUSSIAN Design code to follow. See TR.48.1 Parameter Specifications (on page 2851).</td>
</tr>
<tr>
<td>BCL</td>
<td>15</td>
<td>Compression class of concrete</td>
</tr>
<tr>
<td>CL</td>
<td>0.05</td>
<td>(Slabs/Walls) Distance from top/bottom face of slab/wall element to the center of the longitudinal reinforcing bars located in the first local (x) direction (i.e., main thickness of top/bottom concrete cover for slab/wall elements)</td>
</tr>
<tr>
<td>CL1</td>
<td>0.05</td>
<td>(Beams) Distance from the top or bottom edge of the beam cross section to the center of longitudinal reinforcement (Columns) Distance from the edge of the column cross section to the center of the longitudinal reinforcing bars</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CL2</td>
<td>0.05</td>
<td>(Beams) Distance from left/right side of the beam cross section to the center of longitudinal reinforcement</td>
</tr>
<tr>
<td>CRA</td>
<td>0.05</td>
<td>(Slabs/Walls) Distance from top/bottom face of slab/wall element to the center of the transverse reinforcing bars located in the second local (y) direction (i.e., secondary thickness of top/bottom concrete cover for slab/wall elements)</td>
</tr>
<tr>
<td>DD1</td>
<td>16</td>
<td>(Beams) Diameter of the longitudinal reinforcement bars in the beam tension zone</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Columns) Minimum diameter of longitudinal reinforcement bars for the column</td>
</tr>
<tr>
<td>DD2</td>
<td>16</td>
<td>(Beams) Diameter of the shear reinforcement bars for the beam</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Columns) Maximum diameter of longitudinal reinforcement bars for the column</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>(Beams) Design depth of beam the section. Used for design only, with the default value taken as the YD dimension in the member properties.</td>
</tr>
<tr>
<td>EFA</td>
<td>0</td>
<td>(Beams) Face of support location at the end of the beam</td>
</tr>
<tr>
<td>ELY</td>
<td>1</td>
<td>(Columns) Length coefficient to evaluate slenderness effects in the local Y axis</td>
</tr>
<tr>
<td>ELZ</td>
<td>1</td>
<td>(Columns) Length coefficient to evaluate slenderness effects in the local Z axis</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>FWB</td>
<td>ZB</td>
<td>(Beams) Design width of the beam's bottom flange. Used for design only, with the default value taken as the ZB dimension in the member properties.</td>
</tr>
<tr>
<td>FWT</td>
<td>ZD</td>
<td>(Beams) Design width of the beam's top flange. Used for design only, with the default value taken as the ZD dimension in the member properties.</td>
</tr>
</tbody>
</table>
| MMA            | 0             | (Slabs/Walls) Design parameter of slab/wall reinforcement:  
0. reinforcement calculation applied by stresses in local axis  
1. reinforcement calculation applied by principal stresses |
| MMB            | 1             | (Slabs/Walls) Design parameter of slab/wall reinforcement:  
0. the effect of additional eccentricity is not considered  
1. the effect of additional eccentricity is considered |
| NLT            | 1             | Load case ID number of the long-term load case |
| NSE            | 13            | (Beams) Number of equally spaced sections used for beam design. The upper limit is 20. |
| RCL            | 3             | Class of longitudinal reinforcement:  
1. A240  
2. A400  
3. A500  
4. A600 |
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| RSH            |               | (Beams) Class of shear reinforcement:  
|                |               | 1. A240  
|                |               | 2. A400  
|                |               | 3. A500  
|                |               | 4. A600  |
| SDX            | 16            | (Slabs/Walls) Diameter of reinforcing bars located in the first local (X) direction |
| SDY            | 16            | (Slabs/Walls) Diameter of reinforcing bars located in the second local (Y) direction |
| SELX           | 0             | (Slabs/Walls) Design length of wall member to evaluate slenderness effects in local X axis |
| SELY           | 0             | (Slabs/Walls) Design length of wall member to evaluate slenderness effects in local Y axis |
| SFA            | 0             | (Beams) Face of support location at the start of the beam |
| SSE            |               | (Beam) Limit state parameter for beam design:  
|                |               | 0. reinforcement amount calculated for required load capacity (i.e., the first limit state)  
<p>|                |               | 1. reinforcement amount calculated for cracking requirements (i.e., the second limit state) |</p>
<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| STA            | 0             | (Slabs/Walls) Parameter of limit state for slab/wall design:  
0. non symmetric reinforcement amount calculated for required load capacity (i.e., the first limit state)  
1. symmetric reinforcement amount calculated for required load capacity (i.e., the first limit state)  
2. non symmetric reinforcement amount calculated for cracking requirements (i.e., the second limit state)  
3. symmetric reinforcement amount calculated for cracking requirements (i.e., the second limit state) |
| TEM            | 0             | Parameter of concrete hardening conditions:  
0. normal concrete  
1. fine-grain concrete  
2. (Beams and Columns) fine-grain, steam hardened concrete |
| UB2            | 0.9           | (Slabs/Walls) Specific service conditions coefficient for concrete, $\gamma_b$ |
| UBM            | 1             | Product of service conditions coefficients for concrete, except for $\gamma_b$ in the case of slabs or walls (see UB2) |
| USM            | 1             | (Beams and Columns) Total product of service conditions coefficients for longitudinal reinforcement, $\gamma_s$ |
| WLT            | 0.3           | (Beams and Slabs/Walls) Ultimate width of long-term crack |
| WST            | 0.4           | (Beams and Slabs/Walls) Ultimate width of short-term crack |
### Example

```
STAAD

UNIT MM
MEMBER PROPERTIES
* Columns of rectangular cross-section
  1 TO 16 PRI YD 350. ZD 350.
* Columns of circular cross-section
  17 TO 22 PRI YD 350.
* Beams of T cross-section
  23 TO 40 PRI YD 450. ZD 550. YB 230. ZB 200.
UNIT METER
ELEMENT PROPERTY
  41 TO 100 THICKNESS 0.14
  101 TO 252 THICKNESS 0.16
* Flange of T beams is located at the bottom zone of cross-section
  BETA 180. MEMB 23 TO 40

... * Command of analysis
PERFORM ANALYSIS

... * Output command to print results of calculation (according to user’s judgment)

... * Command of loading and their combinations considered in design
LOAD LIST 1 5 TO 9

* Command to start reinforcement calculation procedure
START CONCRETE DESIGN
CODE RUSSIAN
* List of parameters being used in reinforcement calculation

... * Command of beam reinforcement calculation
  DESIGN BEAM 23 TO 40
* Command of column reinforcement calculation
  DESIGN COLUMN 1 TO 22
* Command of calculation 2D elements (slabs, walls, shells)
  DESIGN ELEMENT 41 TO 252

* Command of interruption of the reinforcement calculation
```

### D13.D.3 Beams

Reinforcement areas are calculated for beams of rectangular and T cross section. When calculating the longitudinal reinforcement bending moment about the local axis Z and torsional moments are considered; the
influence of longitudinal forces and bending moments in relation to local axis Y is ignored. When calculating transverse reinforcement shear forces parallel to local axis Y and torsional moments are considered.

Reinforcement for beams can be calculated either for strength conditions or from crack width limits using the SSE parameter.

In general, the calculation of reinforcement for beams is carried out two times: once according to strength conditions and again according to crack width limitation. In reinforcement calculations for strength conditions (i.e., the first limit state), design load values must be used. In reinforcement calculations for crack width limitation (i.e., the second limit state), characteristic (i.e., normative or service) load values are used. Using the multiple analysis capability of STAAD.Pro allows you to carry out both calculations in a single analysis and design run.

In most cases, calculation of reinforcement is carried out with only a partial number of load cases. In such cases, the LOAD LIST command is used to indicate the load case numbers. The load case number indicated by the NLT parameter – used to indicate the load cases for permanent and long-term loads – must be included in the list of considered loads.

The parameter BCL can be equal to any value of concrete compression strength class given in SNiP 2.03.01-84*, as well as any intermediate value.

It should be noted that the accuracy of results in calculating transverse reinforcement increases with the value of the NSE parameter.

Parameters SFA and EFA are considered only in the calculation of transverse reinforcement. Beam 1 shown in the following figure has rigid end-links of 0.3m at the start of the beam and 0.2m at the end of the beam. In the STAAD input file, this is modeled as:

MEMBER OFFSET
1 START 0.3 0 0
1 END -0.2 0 0

When the MEMBER OFFSET command is used, the forces used in calculation of reinforcement are for a beam whose length is taken from point a to point b (i.e., between the faces of the supporting columns) in the figure. In this case, it is necessary to use the default values of the parameters SFA and EFA (zero).

When the MEMBER OFFSET command not is used, the forces used in calculation of reinforcement are for a beam whose length is taken from node 10 to node 11. In this case, it is necessary to assign the distances to the design parameters SFA and EFA.

CODE RUSSIAN
SFA 0.3 1
EFA 0.2 1
**Note:** The sign is positive for the EFA parameter, as it is assumed to be taken away from the gross beam length.

In both cases, the calculated quantity of transverse reinforcement is the same. The calculated quantity of longitudinal reinforcement in the second case will be greater.

For beams, the following output is generated:

- beam number;
- method of calculation (according to conditions of strength or limitations of opened crack width);
- length and cross-sectional dimensions;
- distance from resultant of forces acting in bottom/top reinforcement to bottom/top edge of the section;
- distance from the side edge of cross-section of the beam web to the centroid of longitudinal bars located at this edge;
- concrete class;
- class of longitudinal and transverse reinforcement;
- assumed in calculations bar diameters of longitudinal and transverse reinforcement;
- calculation results of longitudinal and transverse reinforcement (in two tables).

In nine columns of the first table the following results are presented:

- Section – distance of the section from the “start” of the beam, mm;
- As− – cross-sectional area of longitudinal reinforcement in the bottom zone of cross-section of the beam, if angle BETA=0°, or in the top zone, if BETA=180°, sq.cm;
- As+ – cross-sectional area of longitudinal reinforcement in the top zone of cross-section of the beam, if angle BETA=0°, or in the top zone, if BETA=180°, sq.cm;
- Moments (-/+) – values of bending moments, determining cross-sectional areas of longitudinal reinforcement As* and As*, kNm;
- Load. N. (-/+) – numbers of loading versions, determining cross-sectional areas of longitudinal reinforcement;
- Acrc1 – short-term opened crack width, mm;
- Acrc2 – long-term opened crack width, mm.

Opened crack width is presented only in the case when calculation is performed according to conditions limiting opened crack width.

In ten columns of second table the following results are presented:

- Section – distance of the section from the “start” of the beam, mm;
- Qsw – intensity of transverse reinforcement, kN/m;
- Asw – cross-sectional area of transverse bars, sq.cm, if their step is 10, 15, 20, 25 or 30 cm;
- Q – value of shear force parallel to the local axis, kN;
- T – value of torsional moment, kNm;
- Load N. – number of loading version, determining intensity of transverse reinforcement.

**D13.D.4 Columns**

Reinforcement for columns of rectangular or circular cross section can be calculated. The flexibility of columns can be evaluated in two ways:

- In the case of linear analysis (i.e., the PERFORM ANALYSIS command), the effective length is evaluated using the ELY and ELZ parameters, conforming to provisions of SP5 2.13330.2011.
If a P-DELTA or NONLINEAR (i.e. nonlinear geometry) analysis is performed, the values of the ELY and ELZ parameters should be close to zero (e.g., ELY = ELZ = 0.01).

Longitudinal reinforcement for columns is calculated only for the strength condition. Longitudinal forces and bending moments in relation to local axes Y and Z are taken into account in longitudinal reinforcement calculations.

For rectangular columns, the following output is generated:

- column number;
- column length and cross-sectional dimensions;
- distance of centroid of each longitudinal bar from the nearest edge of the cross-section;
- concrete class;
- longitudinal reinforcement class;
- range of longitudinal reinforcement bar diameters assumed in calculation;
- diameter of longitudinal reinforcement bars obtained in calculation;
- total quantity of longitudinal bars;
- quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis Y;
- quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis Z.

In nine columns of the table under the heading LONGITUDINAL REINFORCEMENT, the following output is presented:

- Section – distance of the section from the "start" of the column, mm;
- Astot – total cross-sectional area of longitudinal reinforcement, sq.cm;
- Asy – cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis Y, sq.cm;
- Asz – cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis Z, sq.cm;
- Percent – reinforcement percentage in the section;
- Nx, Mz, My – respective values of longitudinal force and bending moments in relation to the local axes Z and Y, determining cross-sectional area of longitudinal reinforcement;
- Load.N. – number of loading version, determining cross-sectional area of longitudinal reinforcement.

**D13.D.5 2DElements: Slabs, Walls, and Shells**

In general, the calculation of reinforcement for two dimensional elements (i.e., slabs, walls, and shells) is performed two times: once according to conditions of strength and again according to limiting of width of cracks. In reinforcement calculations for strength conditions (i.e., the first limit state), design load values must be used. In reinforcement calculations for crack width limitation (i.e., the second limit state), characteristic (i.e., normative or service) load values are used. Using the multiple analysis capability of STAAD.Pro allows you to carry out both calculations in a single analysis and design run.

Symmetric or non symmetric reinforcement of elements is calculated according to either first or second limit states by using the STA parameter.

It is necessary to pay close attention to the arrangement of local axis of the element with respect to the direction of reinforcement for the calculation of reinforcement. This is controlled using the CL and CRA parameters, as shown in the following figure.
Design

D. Design Codes
**Design**

D. Design Codes

---

**Figure 198: Russian concrete design parameters CL and CRA**

An example output of reinforcement calculations:

<table>
<thead>
<tr>
<th>Element</th>
<th>Asx</th>
<th>Mx</th>
<th>Nx</th>
<th>Load N.</th>
</tr>
</thead>
<tbody>
<tr>
<td>N.</td>
<td>Asy</td>
<td>My</td>
<td>Ny</td>
<td>Load N. (X)</td>
</tr>
<tr>
<td>sq.cm/m</td>
<td>sq.cm/m</td>
<td>kNm/m</td>
<td>kN/m</td>
<td>kN/m</td>
</tr>
</tbody>
</table>

| 60 TOP  | 0.00 | *4.9 0.0 | 1 | 0.00 | *4.5 0.0 | 1 |
| 61 TOP  | 0.00 | *5.3 0.0 | 1 | 0.00 | *4.7 0.0 | 1 |
| BOT    | 3.53 | *9.9 0.0 | 3 | 3.46 | *8.9 0.0 | 3 |
| BOT    | 3.87 | *10.7 0.0 | 3 | 3.65 | *9.4 0.0 | 3 |

---

STAAD.Pro 2175 User Manual
where:

Element – number of finite element, TOP – “top” zone of member, BOT – “bottom” zone of member (“top” zone of member is determined by positive direction of local axis Z, see the following figure);

Asx – intensity of reinforcing in the first direction (parallel to the local axis X), sq.cm/m;

Mx – distributed bending moment in respect to the local axis Y, kNm/m;

Nx – distributed longitudinal force directed parallel to the axis X, kNm/m;

Load N.(X) – number of loading version, determining intensity of reinforcing in the first direction;

Asy – intensity of reinforcing in the second direction (parallel to the local axis Y), sq.cm/m;

My – distributed bending moment in respect to the local axis X, kNm/m;

Ny – distributed longitudinal force directed parallel to the local axis Y, kN/m;

Load N.(Y) – number of loading version, determining intensity of reinforcing in the second direction.

Figure 199: Local coordinate system of 2D member and notation of forces

D14. South African Codes

D14.A. South African Codes - Concrete Design per SABS-0100-1

STAAD.Pro is capable of performing concrete design based on the South African code SABS-0100-1 2000 Code of Practice for Structural Use of Concrete Part 1: Design. Design can be performed for beams (flexure, shear, and torsion) and columns (axial load + biaxial bending). Given the width and depth (or diameter for circular columns) of a section, the program calculates the required reinforcement.

D14.A.1 Design Parameters

The program contains a number of parameters which are needed to perform and control the design to SABS 0100-1. These parameters not only act as a method to input required data for code calculations but give the engineer control over the actual design process. Default values of commonly used parameters for conventional
Design practice have been chosen as the basis. Table 17A.1 contains a complete list of available parameters with their default values.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 206: South African Concrete Design SABS 0100-1 Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CODE</td>
<td>-</td>
<td>Must be specified as SABS0100. Design code to follow. See <a href="#">TR.5.3.2 Concrete Design-Parameter Specification</a> (on page 2859).</td>
</tr>
<tr>
<td>BRACE</td>
<td>0.0</td>
<td>Column bracing:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Column braced in both directions.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Column braced about local Y direction only</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Column unbraced about local Z direction only</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3. Column unbraced in both Y and Z directions</td>
</tr>
<tr>
<td>CLB</td>
<td>20mm</td>
<td>Clear Cover for outermost bottom reinforcement</td>
</tr>
<tr>
<td>CLS</td>
<td>20mm</td>
<td>Clear Cover for outermost side reinforcement</td>
</tr>
<tr>
<td>CLT</td>
<td>20mm</td>
<td>Clear Cover for outermost top reinforcement</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>Depth of concrete member, in current units. This value default is as provided as YD in MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>ELY</td>
<td>1.0</td>
<td>Member length factor about local Y direction for column design.</td>
</tr>
<tr>
<td>ELZ</td>
<td>1.0</td>
<td>Member length factor about local Z direction for column design.</td>
</tr>
<tr>
<td>FC</td>
<td>30N/mm²</td>
<td>Concrete Yield Stress / cube strength, in current units.</td>
</tr>
</tbody>
</table>
### D14.A.2 Member Dimensions

Concrete members that are to be designed by STAAD.Pro must have certain section properties input under the **MEMBER PROPERTIES** command.

The following example demonstrates the required input:

```
UNIT  MM
MEMBER  PROPERTIES
```

---

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FYMAIN</td>
<td>450 N/mm²</td>
<td>Yield Stress for main reinforcement, in current units.</td>
</tr>
<tr>
<td>FYSEC</td>
<td>450 N/mm²</td>
<td>Yield Stress for secondary reinforcement a, in current units. Applicable to shear bars in beams.</td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>50mm</td>
<td>Maximum required reinforcement bar size. Acceptable bars are per MINMAIN above.</td>
</tr>
<tr>
<td>MINMAIN</td>
<td>8mm</td>
<td>Minimum main reinforcement bar size. Acceptable bar sizes: 6 8 10 12 16 20 25 28 32 36 40 50 60.</td>
</tr>
<tr>
<td>MINSEC</td>
<td>8mm</td>
<td>Minimum secondary bar size a. Applicable to shear reinforcement in beams.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>Output detail</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0. Critical Moment will not be printed with beam design report. Column design gives no detailed results.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. For beam gives min/max steel % and spacing. For columns gives a detailed table of output with additional moments calculated.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Output of TRACK 1.0 List of design sag/hog moments and corresponding required steel area at each section of member.</td>
</tr>
<tr>
<td>WIDTH</td>
<td>ZD</td>
<td>Width of concrete member, in current units. This value default is as provided as ZD in MEMBER PROPERTIES.</td>
</tr>
</tbody>
</table>
In the above input, the first set of members are rectangular (450mm depth x 300mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 300mm diameter. Note that area (AX) is not provided for these members. If shear area areas (AY & AZ) are to be considered in analysis, the user may provide them along with YD and ZD. Also note that if moments of inertias are not provided, the program will calculate them from YD and ZD. Finally a T section can be considered by using the third definition above.

D14.A.3 Beam Design

Beam design includes flexure, shear and torsion. For all types of beam action, all active beam loadings are scanned to create moment and shear envelopes and locate the critical sections. The total number of sections considered is thirteen. From the critical moment values, the required positive and negative bar pattern is developed. Design for flexure is carried out as per clause no. 4.3.3.4.

Shear design as per SABS 0100 clause 4.3.4 has been followed and the procedure includes computation of critical shear values. From these values, stirrup sizes are calculated with proper spacing. If torsion is present, the program will also consider the provisions of SABS 0100 clause 4.3.5. Torsional reinforcement is separately reported.

A TRACK 2 design output is presented below.

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>FLEXURE (Maxm. Sagging/Hogging moments)</th>
<th>VY</th>
<th>SHEAR</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>84.77 MZ 1</td>
<td>-28.13</td>
<td>4.39</td>
<td>1</td>
</tr>
<tr>
<td>500.0</td>
<td>70.70 MZ 1</td>
<td>-28.13</td>
<td>4.39</td>
<td>1</td>
</tr>
<tr>
<td>1000.0</td>
<td>56.64 MZ 1</td>
<td>-28.13</td>
<td>4.39</td>
<td>1</td>
</tr>
<tr>
<td>1500.0</td>
<td>42.57 MZ 1</td>
<td>-28.13</td>
<td>4.39</td>
<td>1</td>
</tr>
<tr>
<td>2000.0</td>
<td>28.50 MZ 1</td>
<td>-28.13</td>
<td>4.39</td>
<td>1</td>
</tr>
<tr>
<td>2500.0</td>
<td>14.43 MZ 1</td>
<td>-28.13</td>
<td>4.39</td>
<td>1</td>
</tr>
</tbody>
</table>
### SUMMARY OF REINF. AREA FOR FLEXURE DESIGN (Sq.mm)

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>TOP</th>
<th>BOTTOM</th>
<th>STIRRUPS</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>543.40/549.78 (7-10í)</td>
<td>680.71/706.86 (9-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>500.0</td>
<td>543.40/549.78 (7-10í)</td>
<td>567.75/603.18 (3-16í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>1000.0</td>
<td>543.40/549.78 (7-10í)</td>
<td>454.79/471.24 (6-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>1500.0</td>
<td>543.40/549.78 (7-10í)</td>
<td>353.21/392.70 (5-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>2000.0</td>
<td>543.40/549.78 (7-10í)</td>
<td>353.21/392.70 (5-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>2500.0</td>
<td>543.40/549.78 (7-10í)</td>
<td>353.21/392.70 (5-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>3000.0</td>
<td>543.40/549.78 (7-10í)</td>
<td>353.21/392.70 (5-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>3500.0</td>
<td>353.21/392.70 (5-10í)</td>
<td>543.40/549.78 (7-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>4000.0</td>
<td>353.21/392.70 (5-10í)</td>
<td>543.40/549.78 (7-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>4500.0</td>
<td>353.21/392.70 (5-10í)</td>
<td>543.40/549.78 (7-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>5000.0</td>
<td>448.91/452.40 (4-12í)</td>
<td>543.40/549.78 (7-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>5500.0</td>
<td>561.87/565.50 (5-12í)</td>
<td>543.40/549.78 (7-10í)</td>
<td>8í @ 115 mm</td>
</tr>
<tr>
<td>6000.0</td>
<td>674.83/678.60 (6-12í)</td>
<td>543.40/549.78 (7-10í)</td>
<td>8í @ 115 mm</td>
</tr>
</tbody>
</table>

**TORSION REINFORCEMENT : Not required**

### D14.A.4 Column Design

Columns are designed for axial force and biaxial bending at the ends. All active loadings are tested to calculate reinforcement. The loading which produces maximum reinforcement is called the critical load and is displayed. The requirements of SABS 0100-1 clause 4.7 are followed, with the user having control on the effective length in each direction by using the ELZ and ELY parameters as described in table 12A.1. Bracing conditions are controlled by using the BRACE parameter. The program will then decide whether or not the column is short or slender and whether it requires additional moment calculations. For biaxial bending, the recommendations of 4.7.4.4 of the code are considered.

Column design is done for square, rectangular and circular sections. For rectangular and square sections, the reinforcement is always assumed to be arranged symmetrically. This causes slightly conservative results in certain cases.
Using parameter TRACK 1.0, the detailed output below is obtained. TRACK 0.0 would merely give the bar configuration, required steel area and percentage, column size and critical load case.

<table>
<thead>
<tr>
<th>COLUMN NO.</th>
<th>DESIGN RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>M30</td>
<td>Fe450 (Main)</td>
</tr>
<tr>
<td>LENGTH:</td>
<td>3000.0 mm</td>
</tr>
</tbody>
</table>

** GUIDING LOAD CASE: 1 END JOINT: 2 SHORT COLUMN**

** DESIGN FORCES (KNS-MET)**
- DESIGN AXIAL FORCE (Pu): -14.6
- About Z
  - INITIAL MOMENTS: 0.00
  - MOMENTS DUE TO MINIMUM ECC.: 0.28
- SLENDERNESS RATIOS: 7.89 4.20
- ADDITION MOMENTS (Maddz and Maddy): 0.00 0.00
- TOTAL DESIGN MOMENTS: 45.17 9.41

** REQD. STEEL AREA:** 26.99 Sq.mm.
** REQD. CONCRETE AREA:** 1213.61 Sq.mm.
** MAIN REINFORCEMENT:** Provide 4 - 12 dia. (0.17%, 452.40 Sq.mm.) (Equally Distributed)
** TIE REINFORCEMENT:** Provide 8 mm dia. rectangular ties @ 140 mm c/c

** SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)**
- Puz: 25.50
- Muz1: 45.22
- Muy1: 51.48

---

### D14.B. South African Codes - Steel Design per SANS10162-1:1993


#### D14.B.1 General

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.
In the STAAD implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The next few sections describe the salient features of the STAAD implementation of SAB0162-1: 1993. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

**D14.B.2 Analysis Methodology**

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

Refer to TR.37 Analysis Specification (on page 2795) for additional information.

**D14.B.3 Member Property Specifications**

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to G.6 Member Properties (on page 2322).

**D14.B.4 Built-in Steel Section Library**

A steel section library consisting of South African Standards shapes is available for member property specification.

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Refer to G.6.2 Built-In Steel Section Libraries (on page 2325) for additional information.

**D14.B.4.1 I Shapes**

The following example illustrates the specification of I- shapes.

```
1 TO 15 TABLE ST IPE-AA100
```

**D14.B.4.2 H shapes**

Designation of H shapes in STAAD is as follows.

For example,

```
18 TO 20 TABLE ST 152X37UC
```
D14.B.4.3 PG shapes
Designation of PG shapes in STAAD is as follows.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 TO 150 TABLE ST 720X200PG</td>
</tr>
</tbody>
</table>

D14.B.4.4 Channel Sections (C & MC shapes)
C and MC shapes are designated as shown in the following example.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 TABLE ST 127X64X15C</td>
</tr>
</tbody>
</table>

D14.B.4.5 Double Channels
Back to back double channels, with or without spacing between them, are specified by preceding the section designation by the letter D. For example, a back to back double channel section PFC140X60 without spacing in between should be specified as:

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 TO 150 TABLE D PFC140X60</td>
</tr>
</tbody>
</table>

A back-to-back double channel section 140X60X16C with spacing 0.01 unit length in between should be specified as:

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 TO 150 TABLE D 140X60X16C SP 0.01</td>
</tr>
</tbody>
</table>

**Note:** The specification SP after the section designation is used for providing the spacing. The spacing should always be provided in the current length unit.

D14.B.4.6 Angles
To specify angles, the letter L succeeds the angle name. Thus, a 70X70 angle with a 25mm thickness is designated as 70X70X8L. The following examples illustrate angle specifications.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 TO 150 TABLE ST 70X70X8L</td>
</tr>
</tbody>
</table>

Note that the above specification is for "standard" angles. In this specification, the local z-axis (see Fig. 2.6 in the Technical Reference Manual) corresponds to the Y'-Y' axis shown in the CSA table. Another common practice of specifying angles assumes the local y-axis to correspond to the Y'-Y' axis. To specify angles in accordance with this convention, the reverse angle designation facility has been provided. A reverse angle may be specified by substituting the word ST with the word RA. Refer to the following example for details.

<table>
<thead>
<tr>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 TO 150 TABLE RA 45X45X3L</td>
</tr>
</tbody>
</table>

The local axis systems for STANDARD and REVERSE angles are shown in Fig. 2.6 of the STAAD Technical Reference manual.

D14.B.4.7 Double Angles
To specify double angles, the specification ST should be substituted with LD (for long leg back-to-back) or SD (short leg back-to-back). For equal angles, either SD or LD will serve the purpose. Spacing between angles may
be provided by using the word SP followed by the value of spacing (in current length unit) after section designation.

| 100 TO 150 TABLE LD 50X50X3L |
| 3 TABLE LD 40X40X5L SP 0.01 |

The second example above describes a double angle section consisting of 40X40X5 angles with a spacing of 0.01 length units.

**D14.B.4.8 Tees**

Tee sections obtained by cutting W sections may be specified by using the T specification instead of ST before the name of the W shape. For example:

| 100 TO 150 TABLE T IPE-AA180 |

will describe a T section cut from a IPE-AA180 section.

**D14.B.4.9 Rectangular Hollow Sections**

These sections may be specified in two possible ways. Those sections listed in the SAB tables may be specified as follows.

| 100 TO 150 TABLE ST TUB60X30X2.5 |

In addition, any tube section may be specified by using the DT (for depth), WT (for width), and TH (for thickness) specifications. For example:

| 100 TO 150 TABLE ST TUBE TH 3 WT 100 DT 50 |

will describe a tube with a depth of 50mm, width of 100mm, and a wall thickness of 3mm. Note that the values of depth, width and thickness must be provided in current length unit.

**D14.B.4.10 Circular Hollow Sections**

Sections listed in the SAB tables may be provided as follows:

| 100 TO 150 TABLE ST PIP34X3.0CHS |

In addition to sections listed in the SAB tables, circular hollow sections may be specified by using the OD (outside diameter) and ID (inside diameter) specifications.
For example:

100 TO 150 TABLE ST PIPE OD 50 ID 48

will describe a pipe with an outside diameter of 50 length units and inside diameter of 48 length units. Note that the values of outside and inside diameters must be provided in terms of current length unit.

### D14.B.4.11 Example

A sample input file to demonstrate usage of South African shapes:

```plaintext
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 30-Mar-05
END JOB INFORMATION
UNIT METER KN

JOINT COORDINATES
1 0 0 0; 2 9 0 0; 3 0 6 0; 4 3 6 0; 5 6 6 0; 6 9 6 0; 7 0 10.5 0;
8 9 10.5 0; 9 2.25 10.5 0; 10 6.75 10.5 0; 11 4.5 10.5 0; 12 1.5 11.4 0;
13 7.5 11.4 0; 14 3 12.3 0; 15 6 12.3 0; 16 4.5 13.2 0;

MEMBER INCIDENCES
1 1 3; 2 3 7; 3 2 6; 4 6 8; 5 3 4; 6 4 5; 7 5 6; 8 7 12; 9 12 14;
10 14 16; 11 15 16; 12 13 15; 13 8 13; 14 9 12; 15 9 14; 16 11 14;
17 11 15; 18 10 15; 19 10 13; 20 7 9; 21 9 11; 22 10 11; 23 8 10;

MEMBER PROPERTY SAFRICAN
1 TABLE ST IPE-AA100
2 TABLE T IPE120
3 TABLE ST 152X23UC
4 TABLE T 152X23UC
5 TABLE ST 812X200PG
6 TABLE T 812X200PG
7 TABLE ST 178X54X15C
8 TABLE D 178X54X15C
9 TABLE D 178X54X15C SP 0.1
10 TABLE ST 25X25X5L
11 TABLE RA 25X25X5L
12 TABLE LD 25X25X5L
13 TABLE SD 25X25X5L
14 TABLE LD 25X25X5L SP 0.1
15 TABLE SD 25X25X5L SP 0.1
16 TABLE ST TUB40X2.5SHS
17 TABLE ST TUBE TH 0 WT 0 DT 50
18 TABLE ST TUBE TH 0.02 WT 100 DT 50
20 TABLE ST PIP48X2.0CHS
21 TABLE ST PIPE OD 0.5 ID 0.48
PRINT MEMBER PROPERTIES
FINISH
```

### D14.B.5 Section Classification

The SAB specification allows inelastic deformation of section elements. Thus, local buckling becomes an important criterion. Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11.2 and Table 1 of SAB0162-1:1993). This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro determines the section classification...
for the standard shapes and user specified shapes. Design is performed for sections that fall into the category of Class 1, 2, or 3 sections only. Class 4 sections are not designed by STAAD.Pro.

**D14.B.6 Member Resistances**

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members’ unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

All the members are checked against allowable slenderness ratio as per Cl.10.2 of SAB0162-1: 1993.

**D14.B.6.1 Axial**

Parameters FYLD, FU, and NSF are applicable for these calculations.

**D14.B.6.2 Axial Compression**

The compressive resistance of columns is determined based on Clause 13.3 of the code. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (KL/r ratios). The effective length for the calculation of compression resistance may be provided through the use of the parameters KX, KY, KZ, LX, LY, and LZ (see D14.B.7 Design Parameters (on page 2187)). Some of the aspects of the axial compression capacity calculations are:

1. For frame members not subjected to any bending, and for truss members, the axial compression capacity in general column flexural buckling is calculated from Cl.13.3.1 using the slenderness ratios for the local Y-Y and Z-Z axis. The parameters KY, LY, KZ, and LZ are applicable for this.
2. For single angles, asymmetric or cruciform sections are checked as to whether torsional-flexural buckling is critical. But for KL/r ratio exceeding 50, as torsional flexural buckling is not critical, the axial compression capacities are calculated by using Cl.13.3. The reason for this is that the South African code doesn’t provide any clear guidelines for calculating this value. The parameters KY, LY, KZ, and LZ are applicable for this.
3. The axial compression capacity is also calculated by taking flexural-torsional buckling into account. Parameters KX and LX may be used to provide the effective length factor and effective length value for flexural-torsional buckling. Flexural-torsional buckling capacity is computed for single channels, single angles, Tees and Double angles.
4. While computing the general column flexural buckling capacity of sections with axial compression + bending, the special provisions of 13.8.1(a), 13.8.1(b) and 13.8.1(c) are applied. For example, Lambda = 0 for 13.8.1(a), K=1 for 13.8.1 (b), etc.

**D14.B.6.3 Bending**

The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified in STAAD with the help of the parameter UNL. If UNL is less than one tenth the member length (member length is the distance between the joints of the member), the member is treated as being continuously laterally supported. In this case, the moment resistance is computed from Clause 13.5 of the code. If UNL is greater than or equal to one-tenth the member length, its value is used as the laterally unsupported length. The equations of Clause 13.6 of the code are used to arrive at the moment of resistance of laterally unsupported members. Some of the aspects of the bending capacity calculations are:

1. The weak axis bending capacity of all sections except single angles is calculated as:
For Class 1 & 2 sections

\[
\Phi \times Py \times Fy
\]

For Class 3 sections

\[
\Phi \times Sy \times Fy
\]

Where:

- \( \Phi \) = Resistance factor = 0.9
- \( Py \) = Plastic section modulus about the local Y axis
- \( Sy \) = Elastic section modulus about the local Y axis
- \( Fy \) = Yield stress of steel

2. Single angles sections are not designed by STAAD, as the South African code doesn't provide any clear guidelines for calculating this value.

3. For calculating the bending capacity about the Z-Z axis of singly symmetric shapes such as Tees and Double angles, SAB0162-1: 1993 stipulates in Clause 13.6(b), page 31, that a rational method.

**D14.B.6.4 Axial compression and bending**

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. In these equations, the additional bending caused by the action of the axial load is accounted for by using amplification factors. Clause 13.8 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceeds 1.0 or the allowable value provided using the RATIO parameter (see **D14.B.7 Design Parameters** (on page 2187)), the member is considered to have FAILED under the loading condition.

**D14.B.6.5 Axial tension and bending**

Members subjected to axial tension and bending are also designed using interaction equations. Clause 13.9 of the code is used to perform these checks. The actual RATIO is determined as the value of the left hand side of the critical equation.

**D14.B.6.6 Shear**

The shear resistance of the cross section is determined using the equations of Clause 13.4 of the code. Once this is obtained, the ratio of the shear force acting on the cross section to the shear resistance of the section is calculated. If any of the ratios (for both local Y & Z axes) exceed 1.0 or the allowable value provided using the RATIO parameter (see **D14.B.7 Design Parameters** (on page 2187)), the section is considered to have failed under shear. The code also requires that the slenderness ratio of the web be within a certain limit (See Cl. 13.4.1.3, page 29 of SABS 0162-1:1993). Checks for safety in shear are performed only if this value is within the allowable limit. Users may by-pass this limitation by specifying a value of 2.0 for the MAIN parameter.

**D14.B.7 Design Parameters**

The design parameters outlined in table below may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.
Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 207: South African Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEAM</td>
<td>0</td>
<td>0 - Perform design at ends and those locations specified in the section command. 1 - Perform design at ends and 1/12th section locations along member length.</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Greater than 0.0 and less than 2.5, Value of Omega_2 (C1.13.6) to be used for calculation. Equal to 0.0: Calculate Omega_2</td>
</tr>
<tr>
<td>CMY</td>
<td>1.0</td>
<td>1 - Do not calculate Omega-1 for local Y axis. 2 - Calculate Omega-1 for local Y axis</td>
</tr>
<tr>
<td>CMZ</td>
<td>1.0</td>
<td>1 - Do not calculate Omega-1 for local Z axis. 2 - Calculate Omega-1 for local Z axis</td>
</tr>
<tr>
<td>DFF</td>
<td>0</td>
<td>Default is 0 indicating that deflection check is not performed</td>
</tr>
<tr>
<td>DJ1</td>
<td>0</td>
<td>Start node of physical member for determining deflected pattern for deflection check and should be set along with DFF parameter</td>
</tr>
<tr>
<td>DJ2</td>
<td>0</td>
<td>End node of physical member for determining deflected pattern for deflection check and should be set along with DFF parameter</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DMAX</td>
<td>1000</td>
<td>Maximum allowable depth</td>
</tr>
<tr>
<td>DMIN</td>
<td>0</td>
<td>Minimum required depth</td>
</tr>
<tr>
<td>FU</td>
<td>345Mpa</td>
<td>Ultimate strength of steel</td>
</tr>
<tr>
<td>FYLD</td>
<td>300Mpa</td>
<td>Yield strength of steel</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td>K value for flexural torsional buckling</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local Y-axis, usually minor axis</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local Z-axis, usually major axis</td>
</tr>
<tr>
<td>LT</td>
<td>Member length</td>
<td>Length for flexural torsional buckling</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Length in local Y axis for slenderness value KL/r</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Length in local Z axis for slenderness value KL/r</td>
</tr>
<tr>
<td>MAIN</td>
<td>0</td>
<td>Flag for controlling slenderness check</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of applied load to section capacity</td>
</tr>
<tr>
<td>SSY</td>
<td>0</td>
<td>Sidesway parameter</td>
</tr>
</tbody>
</table>

- **MAIN**
  - **0**: For Check for slenderness.
  - **1**: For Do not check for slenderness

- **SSY**
  - **0**: Sideway about local Y-axis.
  - **1**: No sideway about local Y-axis.
### Design Codes

#### D14.B.8 Code Checking

The purpose of code checking is to determine whether the current section properties of the members are adequate to carry the forces obtained from the most recent analysis. The adequacy is checked as per the SANS10162-1:1993 requirements.

Code checking is done using forces and moments at specified sections of the members. If the BEAM parameter for a member is set to 1 (which is also its default value), moments are calculated at every twelfth point along the beam. When no section locations are specified and the BEAM parameter is set to zero, design will be based on member start and end forces only. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. Using the TRACK parameter can control the extent of detail of the output.

Refer to [D1.B.1.3 Code Checking](on page 1418) for general information on Code Checking. Refer to [TR.49 Code Checking Specification](on page 2852) for details the specification of the Code Checking command.

---

#### Example

Sample input data for South African Code Design

```plaintext
PARAMETER
CODE  SANS10162-1: 1993
MAIN  1 all
LY  4 MEMB 1
LZ  4 MEMB 1
UNL 4 MEMB 1
CB 0 MEMB 1 TO 23
```
D14.B.9 Member Selection

The member selection process involves determination of the least weight member that passes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially.

For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on members listed as PRISMATIC.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.

D14.B.10 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. The term CRITICAL COND refers to the section of the SANS10162-1:1993 specification, which governed the design.

If the TRACK parameter is set to 1.0, the output will be displayed as follows:

```
**************************************
STAAD.PRO CODE CHECKING
(SOUTHAFRICAN STEEL/SANS10162-1:1993)
**************************************
ALL UNITS ARE - KNS MET (UNLESS OTHERWISE NOTED)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
               FX            MY             MZ       LOCATION
=======================================================================
1 ST   406X67UB                 (SOUTHAFRICAN SECTIONS)
PASS       SAB-13.8           0.543         1
0.00            0.00        -191.90        4.08
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>FACTORED RESISTANCES FOR MEMBER-     1  UNIT - KN,M    PHI = 0.90</td>
</tr>
<tr>
<td>MRZ=      353.27  MRY=       63.99</td>
</tr>
<tr>
<td>CR=       453.21  TR=      2388.50  VR=       642.00</td>
</tr>
<tr>
<td>---------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
```
Factored member resistances will be printed out. Following is a description of some of the items printed out.

<table>
<thead>
<tr>
<th>Output Term</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MRZ</td>
<td>Factored moment of resistance in z direction</td>
</tr>
<tr>
<td>MRY</td>
<td>Factored moment of resistance in y direction</td>
</tr>
<tr>
<td>CR</td>
<td>Factored compressive resistance for column</td>
</tr>
<tr>
<td>TR</td>
<td>Factored tensile capacity</td>
</tr>
<tr>
<td>VR</td>
<td>Factored shear resistance</td>
</tr>
</tbody>
</table>

Further details can be obtained by setting TRACK to 2.0. A typical output of track 2.0 parameter is as follows.

```
***************************************
STAAD.PRO CODE CHECKING
(SOUTHAFRICAN STEEL/SANS10162-1:1993 )
***************************************
ALL UNITS ARE - KNS MET (UNLESS OTHERWISE NOTED)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/     LOCATION
FX            MY             MZ       LOCATION
=======================================================================
1 ST   406X67UB                 (SOUTHAFRICAN SECTIONS)
PASS       SAB-13.8           0.543         1
0.00            0.00        -191.90        4.08

MEMBER PROPERTIES (UNIT = CM)
-----------------------------
CROSS SECTION AREA =  8.55E+01   MEMBER LENGTH =  7.00E+02
IZ =  2.43E+04   SZ =  1.19E+03   PZ =  1.35E+03
IY =  1.36E+03   SY =  1.52E+02   PY =  2.37E+02

MATERIAL PROPERTIES (UNIT = MPA)
--------------------------------
FYLD = 300.0   FU = 345.0

SECTION CAPACITIES (UNIT - KN,M)
--------------------------------
CRY =  4.532E+02   CRZ =  2.016E+03
CTORFLX = 4.532E02
TENSILE CAPACITY = 2.308E+03  COMpressive capacity = 4.532E+02
FACTORED MOMENT RESISTANCE : MRY = 6.399E+01   MRZ = 3.533E+02
FACTORED SHEAR RESISTANCE  : VRY = 6.420E+02   VRZ = 6.075E+02

MISCELLANEOUS INFORMATION
```
Following is a description of some of the items printed out.

<table>
<thead>
<tr>
<th>Output Term</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CRY</td>
<td>Factored compressive resistance for column buckling about the local y axis</td>
</tr>
<tr>
<td>CRZ</td>
<td>Factored compressive resistance for column buckling about the local z axis</td>
</tr>
<tr>
<td>CTORFLX</td>
<td>Factored compressive resistance against torsional flexural buckling</td>
</tr>
<tr>
<td>TENSILE CAPACITY</td>
<td>Factored tensile capacity</td>
</tr>
<tr>
<td>COMPRESSIVE CAPACITY</td>
<td>Factored compressive capacity</td>
</tr>
</tbody>
</table>
| FACTORED MOMENT RESISTANCE | MRY = Factored moment of resistance in y direction  
MRZ = Factored moment of resistance in z direction |
| FACTORED SHEAR RESISTANCE | VRY = Factored shear resistance in y direction  
VRZ = Factored shear resistance in z direction |

D14.B.11 Verification Examples

In the next few pages are included three verification examples for reference purposes.

**D14.B.11.1 Verification Example No. 1**

Determine the capacity of a South African I-section column in axial compression per South African steel design code (SANS10162-1:1993). Column is braced at its ends for both axes.

**Reference**


**Given**

FYLD = 300 Mpa

Length = 6,000 mm

**Comparison**
## Table 208: SABS 0162-1:1993 Verification problem no.1 comparison

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Compressive Strength (kN)</td>
<td>1,516</td>
<td>1,516</td>
<td>none</td>
</tr>
</tbody>
</table>

### Input File

```
STAAD PLANE
START JOB INFORMATION
* ENGINEER DATE
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY SAFRICAN
1 TABLE ST 356X67UB
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+008
POISSON 0.3
DENSITY 76.8191
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 248210 FU 399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL ALL
UNIT METER KN
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FY -1500
PERFORM ANALYSIS
PARAMETER 1
CODE SANS10162-1: 1993
LZ 6 ALL
LY 3 ALL
FU 450000 ALL
BEAM 1 ALL
NSF 0.85 ALL
TRACK 2 ALL
FYLD 300000 ALL
CHECK CODE ALL
FINISH
```

### Output

```
***********************************************************************
STAAD.PRO CODE CHECKING
***********************************************************************
```
### D14.B.11.2 Verification Example No. 2

Determine the capacity of a South African I-section beam in bending per South African steel design code (SANS10162-1:1993). The beam has torsional and simple lateral rotational restraint at the supports, and the

---

**MEMBER PROPERTIES**

**UNIT = CM**

<table>
<thead>
<tr>
<th>SECTION CLASS</th>
<th>CROSS SECTION AREA</th>
<th>MEMBER LENGTH</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8.55E+01</td>
<td>6.00E+02</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>INERTIA</th>
<th>CROSS AXES</th>
<th>SHEAR AXES</th>
</tr>
</thead>
<tbody>
<tr>
<td>IZ</td>
<td>1.95E+04</td>
<td>PZ</td>
</tr>
<tr>
<td>SY</td>
<td>1.57E+02</td>
<td>PY</td>
</tr>
</tbody>
</table>

**MATERIAL PROPERTIES**

**UNIT = MPA**

<table>
<thead>
<tr>
<th>FLOW</th>
<th>TENSILE</th>
</tr>
</thead>
<tbody>
<tr>
<td>300.0</td>
<td>450.0</td>
</tr>
</tbody>
</table>

**SECTION CAPACITIES**

**UNIT = KN,M**

<table>
<thead>
<tr>
<th>CRY</th>
<th>CRZ</th>
<th>CTORFLX</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.516E+03</td>
<td>2.038E+03</td>
<td>1.516E+03</td>
</tr>
</tbody>
</table>

**FACTORED RESISTANCE**

<table>
<thead>
<tr>
<th>MRY</th>
<th>MRZ</th>
<th>VRY</th>
<th>VRZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.561E+01</td>
<td>3.267E+02</td>
<td>5.903E+02</td>
<td>6.461E+02</td>
</tr>
</tbody>
</table>

**MISCELLANEOUS INFORMATION**

**NET SECTION FACTOR FOR TENSION**

<table>
<thead>
<tr>
<th>KL/RY</th>
<th>KL/RZ</th>
<th>ALLOWABLE KL/R</th>
</tr>
</thead>
<tbody>
<tr>
<td>75.220</td>
<td>39.730</td>
<td>200.000</td>
</tr>
</tbody>
</table>

---
applied point load provides effective lateral restraint at the point of application is braced at its ends for both axes.

Reference


Given

FYLD = 300 Mpa

Comparison

Table 209: SAB 0162 -1:1993 Verification Problem 2 comparison

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major Axis Bending Resistance (kN)</td>
<td>353.4</td>
<td>364.5</td>
<td>3%</td>
</tr>
</tbody>
</table>

Input File

```
STAAD PLANE
START JOB INFORMATION
*ENGINEER DATE
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 7 0 0
MEMBER INCIDENCES
1 1 3; 2 3 2
MEMBER PROPERTY SAFRICAN
1 2 TABLE ST 406X67UB
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.00E+008
POISSON 3
DENSITY 977
ISOTROPIC STEEL
E 2.00E+008
POISSON 3
DENSITY 8195
ALPHA 2E-005
DAMP 03
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL MEMB 1 2
UNIT METER KN
SUPPORTS
1 3 PINNED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -104 4
1 UNI GY -4
2 UNI GY -2
PERFORM ANALYSIS
```
PARAMETER
CODE SANS10162-1: 1993
CB 0 ALL
UNL 4 MEMB 1
FU 450000 ALL
BEAM 1 ALL
NSF 85 ALL
FYLD 300000 ALL
TRACK 2 ALL
CHECK CODE MEMB 1
FINISH

Output

******************************************************************************
STAAD.PRO CODE CHECKING
(SOUTHAFRICAN STEEL/SANS10162-01:1993)
******************************************************************************

ALL UNITS ARE - KNS MET (UNLESS OTHERWISE Noted)

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>1 ST</td>
<td>406X67UB</td>
<td>PASS</td>
<td>SAN-13.8</td>
<td>0.526</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>-191.90</td>
<td>4.08</td>
</tr>
</tbody>
</table>

MEMBER PROPERTIES (UNIT = CM)

SECTION CLASS = 1
CROSS SECTION AREA = 8.55E+01  MEMBER LENGTH = 7.00E+02
IZ = 2.43E+04  SZ = 1.19E+03  PZ = 1.35E+03
IY = 1.36E+03  SY = 1.52E+02  PY = 2.37E+02

MATERIAL PROPERTIES (UNIT = MPA)

FYLD = 300.0  FU = 450.0

SECTION CAPACITIES (UNIT - KN,M)

CRY = 4.532E+02  CRZ = 2.016E+03
CTORFLX = 4.532E+02
TENSILE CAPACITY = 2.308E+03  COMpressive CAPACITY = 4.532E+02
FACTORED MOMENT RESISTANCE : MRY = 6.399E+01  MRZ = 3.645E+02
FACTORED SHEAR RESISTANCE : VRY = 6.420E+02  VRZ = 6.075E+02
MISCELLANEOUS INFORMATION
--------------------------
NET SECTION FACTOR FOR TENSION = 85.000
UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) = 4.000
OMEGA-1 (Y-AXIS) = 1.00  OMEGA-1 (Z-AXIS) = 1.00  OMEGA-2 = 1.75
SHEAR FORCE (KNS) : Y AXIS = -6.305E+01  Z AXIS = 0.000E+00
SLENDERNESS RATIO OF WEB (H/W) = 4.33E+01
KL/RY = 175.514  KL/RZ = 41.522  ALLOWABLE KL/R = 300.000

D14.B.11.3 Verification Example No. 3

Determine the elastic shear capacity per South African steel design code (SANS10162-1:1993) of a South African I-section which is simply supported over the span of 8 m.

Reference

Given
FYLD = 300 Mpa

Comparison

Table 210: SAB 0162-1:1993 Verification Problem 3 comparison

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear Capacity (kN)</td>
<td>687.1</td>
<td>687.1</td>
<td>none</td>
</tr>
</tbody>
</table>

Input File

STAAD PLANE
START JOB INFORMATION
*ENGINEER DATE
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 8 0 0
MEMBER INCIDENCES
1 1 2
MEMBER PROPERTY SAFRICAN
1 TABLE ST 457X67UB
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2E+008
POISSON 3
DENSITY 977
ISOTROPIC STEEL
E 2E+008
POISSON 3
DENSITY 8195
ALPHA 2E-005
DAMP 03
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL MEMB 1
UNIT METER KN
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -70
PERFORM ANALYSIS
PARAMETER
CODE SANS10162-1: 1993
FU 450000 ALL
BEAM 1 ALL
FYLD 300000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

Output

*****************************
STAAD.PRO CODE CHECKING
(SOUTHAFRICAN STEEL/SANS10162-01:1993 )
*****************************

ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)

MEMBER TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
FX            MY             MZ       LOCATION
=======================================================================
*     1 ST   457X67UB                 (SOUTHAFRICAN SECTIONS)
FAIL       SAN-13.8           1.411         1
0.00            0.00        -560.00        4.00

MEMBER PROPERTIES (UNIT = CM)
-----------------------------
SECTION CLASS =   1
CROSS SECTION AREA =  8.55E+01   MEMBER LENGTH =  8.00E+02
IZ =  2.94E+04   SZ =  1.30E+03   PZ =  1.47E+03
IY =  1.45E+03   SY =  1.53E+02   PY =  2.37E+02

MATERIAL PROPERTIES (UNIT = MPA)
--------------------------------
FYLD = 300.0   FU = 450.0

SECTION CAPACITIES (UNIT - KN,M)

---

STAAD.Pro is capable of performing steel design based on the South African code SANS 10162-1:2011 Design of steel structures.

D14.C.1 General

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability, and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The next few sections describe the salient features of the STAAD implementation of SANS 10162-1:2011. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D14.C.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

Refer to TR.37 Analysis Specification (on page 2795) for additional information.
D14.C.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to G.6 Member Properties (on page 2322).

D14.C.4 Built-in Steel Section Library

A steel section library consisting of South African Standards shapes is available for member property specification.

Refer to D14.B.4 Built-in Steel Section Library (on page 2182) for details on using the built-in library of South African steel shapes.

D14.C.5 Section Classification

The SANS 10162 specification allows inelastic deformation of section elements. Thus, local buckling becomes an important criterion. Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11.2 and Table 1 of SANS 10162-1:2011). Class 4 sections are those with ratios exceeding those listed for Class 3. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro determines the section classification for the standard shapes and user specified shapes. Design is performed accordingly for sections in any category (including slender elements).

D14.C.6 Member Resistances

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members’ unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

All the members are checked against allowable slenderness ratio as per Cl.10.2 of SANS 10162-1:2011.

D14.C.6.1 Resistance Factor

The strength resistance factor, \( \phi \), used for structural steel is 0.90 per Section 13.1. This factor is applied to the equations directly in capacity calculations.

D14.C.6.2 Axial Tension

The factored tensile resistance, \( T_r \), developed by a member subjected to an axial tensile force shall be taken as least of the following per Section 13.2:

\[
T_r = \phi A_{fg} f_y \\
T_r = 0.85 \phi A_{ne} f_u
\]

Parameters FYLD, FU, and NSF are applicable for these calculations.
**D14.C.6.3 Flexural Buckling - Axial Compression**

Per section 13.3.1, the factored axial compressive resistance, $C_r$, of doubly symmetric sections conforming to the requirements of clause 11 for class 1, 2, or 3 sections shall be taken as:

$$C_r = \phi A f_y (1 + \lambda^{2n})^{-1/n}$$

where

- $n = 1.34$ for hot-rolled, fabricated structural sections, and hollow structural sections manufactured according to SANS 657-1
- $\lambda = \frac{kl}{r} \sqrt{\frac{f}{\pi E}} = \sqrt{\frac{f}{f_y}}$

Doubly symmetric sections which may be governed by torsional-flexural buckling shall also meet the requirements of 13.3.2.

The parameters $K_Y$, $L_Y$, $K_Z$, and $L_Z$ are applicable for this.

**D14.C.6.4 Torsional or Torsional-Flexural Buckling**

Per section 13.3.2, the factored compressive resistance, $C_r$, of asymmetric, singly symmetric, and cruciform or other bisymmetric sections not covered under 13.3.1 shall be computed using the expressions given in 13.3.1 with a value of $n = 1.34$ and the value of $f_e$ taken as lesser of $F_{ex}$ and $F_{eyz}$ for single symmetric section, with the $y$ axis taken as the axis of symmetry.

$$f_{eyz} = \frac{f_{ey} + f_{ez}}{2\Omega} \left[ 1 - \sqrt{1 - \frac{4f_{ey}f_{ez}^2}{(f_{ey} + f_{ez})^2}} \right]$$

where

- $f_{ey} = n^2E \left( K_y L_y r_y \right)^2$
- $f_{ez} = \left( n^2 E C_w + GJ \right) \frac{1}{A r_o^2}$
- $\Omega = 1 - \left( \frac{x_0^2 + y_0^2}{\bar{r}_0^2} \right)$
- $x_0, y_0 = \text{the principal coordinates of the shear center with respect to the centroid of the cross-section.}$
- $\bar{r}_0^2 = x_0^2 + y_0^2 + r_x^2 + r_y^2$

For asymmetric sections, $f_e$ is the smallest root of:

$$\left( f_e - f_{ex} \right) \left( f_e - f_{ey} \right) \left( f_e - f_{ez} \right) - f_e^2 \left( f_e - f_{ey} \right) \left( \frac{x_0}{r_0} \right)^2 - f_e^2 \left( f_e - f_{ez} \right) \left( \frac{y_0}{r_0} \right)^2 = 0$$

where

- $f_{ex} = n^2E \left( K_x L_x r_x \right)^2$

The parameters $K_Y$, $L_Y$, $K_Z$, and $L_Z$ are applicable for this.
D14.C.6.5 Shear

Per section 13.4.1, the factored shear resistance, \( V_r \), developed by the web of a flexural member, shall be taken as

\[ V_r = \phi A_v f_s \]

where

\[ A_v = \text{the shear area} \quad t_w h \]

a. \( f_s = 0.66f_y \) when \( \frac{h_w}{t_w} \leq 440 \sqrt{\frac{k_v}{f_y}} \)

where

\[ k_v = \text{the shear buckling coefficient defined as:} \]

\[ = 4 + \frac{5.34}{(s/h_w)^2} \text{ if } s \leq h_w < 1 \]

\[ = 5.34 + \frac{4}{(s/h_w)^2} \text{ if } s \geq h_w \geq 1 \]

b. \( f_s = f_{cri} \) when \( 440 \sqrt{\frac{k_v}{f_y}} < \frac{h_w}{t_w} \leq 500 \sqrt{\frac{k_v}{f_y}} \)

where

\[ f_{cri} = \frac{290 \sqrt{f_y h_w}}{h_w / t_w} \]

\[ k_v = \text{is as defined in (a)} \]

c. \( f_s = f_{cri} + f_t \) when \( 500 \sqrt{\frac{k_v}{f_y}} < \frac{h_w}{t_w} \leq 620 \sqrt{\frac{k_v}{f_y}} \)

where

\[ f_t = \text{the tension field post-buckling stress defined as:} \]

\[ = k_a(0.50f_y - 0.866f_{cri}) \]

\[ k_a = \text{the aspect coefficient defined as:} \]

\[ = \frac{1}{1 + (s/h_w)^2} \]

d. \( f_s = f_{cre} + f_t \) when \( 620 \sqrt{\frac{k_v}{f_y}} < \frac{h_w}{t_w} \)

where

\[ f_t = \text{the tension field post-buckling stress defined as:} \]

\[ = k_a(0.50f_y - 0.866f_{cre}) \]

\[ f_{cre} = 180,000k_v \]

\[ k_v = \text{as defined in (a)} \]

D14.C.6.6 Bending

The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified using the parameters UNB and UNT.

Laterally Supported Members
Per section 13.5, the factored moment resistance, $M_r$, developed by a member subjected to uniaxial bending moments about a principal axis and where continuous lateral support is provided to the compressive flange shall be taken as:

a. For class 1 and class 2 sections:

$$M_r = \phi Z_p f_y = \phi M_p$$

b. For class 3 sections:

$$M_r = \phi Z_e f_y = \phi M_y$$

Laterally Unsupported Members

Per section 13.6, where continuous lateral support is not provided to the compression flange of a member subjected to uniaxial strong axis bending, the factored moment resistance, $M_r$, may be taken as follows:

a. For doubly symmetric class 1 and class 2 sections, except closed square and circular sections:

i. When $M_{cr} > 0.67M_p$,

$$M_r = 1.15\phi M_p \left(1 - \frac{0.28M_p}{M_{cr}}\right)$$

but not greater than $\phi M_p$.

ii. When $M_{cr} \leq 0.67M_p$,

$$M_r = \phi M_{cr}$$

where

$$M_{cr} = \omega_2^2 \pi^2 KL E I_y GJ + \left( \frac{\pi E}{KL} \right)^2 I_y C_w$$

$KL$ = the effective length of the unbraced portion of the beam, in mm.

$\omega_2$ =

- $\omega_2^2 = 1.75 + 1.05\kappa + 0.3\kappa^2 \leq 2.5$ for unbraced lengths subjected to end moments, or
- $\omega_2^2 = 1.0$ when the bending moment at any point within the unbraced length is larger than the end moment or when there is no effective lateral support for the compression flange at one of the ends of the unsupported length.

Note: The value for $\omega_2$ can be specified using the CB parameter. Otherwise, it is calculated as indicated here.

$\kappa$ = the ratio of the smaller factored moment to the larger factored moment at opposite ends of the unbraced length, positive for double curvature and negative for single curvature.

$C_w$ = the ratio of the smaller factored moment to the larger factored moment at opposite ends of the unbraced length, positive for double curvature and negative for single curvature.

Note: Alternatively, $E$ may be specified directly.

b. For doubly symmetric class 3 sections, except closed square and circular sections, and for channels:

i. When $M_{cr} > 0.67M_p$, 

$$M_r = \phi M_{cr}$$
Design
D. Design Codes

\[ M_r = 1.15 \phi M_y \left( 1 - \frac{0.28 M_y}{M_{cr}} \right) \]

but not greater than \( \phi M_y \) for class 3 sections and the value given in 13.5(c)(iii) for class 4 sections.

**ii.** When \( M_{cr} \leq 0.67 M_y \),

\[ M_r = \phi M_{cr} \]

where \( M_{cr} \) and \( \omega_2 \) are as defined in 13.6(a).

**c.** For closed sections and circular sections, \( M_{cr} \) shall be determined in accordance with section 13.5.

**d.** For biaxial bending, the member shall meet the following criterion:

\[ \frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \leq 1.0 \]

**e.** For monosymmetric sections, a rational method of analysis should be used.

**Note:** STAAD.Pro uses AISC LRFD guidelines for the design of channels, double angles, tees, and single angle sections.

**i.** For tees and double angles:

\[ = \frac{\pi \sqrt{E J G J}}{L_b} \left( B + \sqrt{1 + B^2} \right) \]  
(AISC LRFD equation F1-15)

where

\[ B = -2.3(d / UNL)Jy/J \]

minus sign considered for conservative side

**ii.** For channel sections:

When \( M_{cr} \leq 0.67 M_y \):

\[ M_r = 1.15 \phi M_y \left( 1 - 0.28 \frac{M_y}{M_{cr}} \right) \leq 0.9 M_y \]

When \( M_{cr} > 0.67 M_y \)

\[ M_r = 0.9 M_{cr} \]

**iii.** For angle sections:

When \( M_{ob} \leq M_y \):

\[ M_n = M_{ob} \left[ 0.92 - 0.17 \frac{M_{ob}}{M_y} \right] \]

When \( M_{ob} > M_y \)

\[ M_n = M_y \left[ 1.92 - 1.17 \sqrt{\frac{M_y}{M_{ob}}} \right] \leq 1.5 M_y \]

where

\[ M_{ob} = 4.9E \frac{l^2}{t^2} CB \left[ \sqrt{\beta_w^2 + 0.052(lt / rz)^2 + \beta_w} \right] \text{ for unequal leg angles.} \]

**Note:** \( \beta_w \) is conservatively assumed as zero.
\[ I_2 = \frac{CB \cdot 0.46Eb^2}{t^2} \] for equal leg angles.

- \( I_2 \) = minor principal axis moment of inertia
- \( r_2 \) = radius of gyration for minor principal axis
- \( \beta_w \) = 0 (conservative assumption)
- \( CB \) = The design parameter corresponding to \( \omega_2 \).
- \( b \) = width of the angle leg
- \( t \) = thickness of the angle

### D14.C.6.7 Member Strength and Stability

**Class 1 and Class 2 I-Shaped Sections**

Members required to resist both bending moments and an axial compressive force shall be proportioned so that:

\[
\frac{C_u}{C_r} + \frac{0.85U_{1x}M_{ux}}{M_{rx}} + \frac{\beta U_{1y}M_{uy}}{M_{ry}} \leq 1.0
\]

where

- \( C_u \) and \( M_u \) = the maximum load effects in compression and bending, respectively, including stability effects as defined in Cl. 8.7.
- \( \beta = 0.6 + 0.4\lambda_y \leq 0.85 \)
- \( \lambda_y \) = the nondimensional slenderness parameter about the y-y axis

The capacity of the member shall be examined for:

**a.** cross-sectional strength (members in braced frames only), with \( \beta = 0.6 \), in which case:

- \( C_r \) is as defined in Cl. 13.3 with the value of \( \lambda = 0 \).
- \( M_r \) is as defined in Cl. 13.5 (for the appropriate class of section), and
- \( U_{1x} \) and \( U_{1y} \) are as defined in 13.8.4 but not less than 1.0, and

**b.** overall member strength, in which case:

- \( C_r \) is as defined in Cl. 13.3 with the value of \( K = 1 \), except that for strong-axis bending, \( C_r = C_{rx} \) (see also 10.3.2),
- \( M_r \) is as defined in Cl. 13.5 (for the appropriate class of section), and
- \( U_{1x} \) and \( U_{1y} \) are as defined in 13.8.4 for members in braced frames, and
- \( U_{1x} \) and \( U_{1y} \) are taken as 1.0 for members in unbraced frames, and

**c.** lateral torsional bucking strength, when applicable, in which case:

- \( C_r \) is as defined in Cl. 13.3, and is based on weak-axis or torsional-flexural buckling (see also 10.3.3),
- \( M_{rx} \) is as defined in 13.6 (for the appropriate class of section),
- \( M_{ry} \) is as defined in 13.5 (for the appropriate class of section),
- \( U_{1x} \) and \( U_{1y} \) are as defined in 13.8.4 for members in braced frames, and
- \( U_{1x} \) is as defined in 13.8.4 but not less than 1.0, for members in braced frames, and
- \( U_{1y} \) is as defined in 13.8.4 for members in braced frames

There parameters \( SSY \) and \( SSZ \) are used to indicate the sidesway in the local Y and Z axes, respectively.

In addition, the member shall meet the following criteria:

\[
\frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \leq 1.0
\]
where

\[ M_{rx} \text{ and } M_{ry} = \text{ as described in 13.5 or 13.6, as appropriate} \]

All Other Sections

Members required to resist both bending moments and an axial compressive force shall be proportioned so that:

\[ \frac{C_u}{C_r} + \frac{U_1 M_{ux}}{M_{rx}} + \frac{U_1 M_{uy}}{M_{ry}} \leq 1.0 \]

where all terms are as defined in 13.8.2

The capacity of the member shall be examined for the following cases in a parallel manner to that in 13.8.2:

a. cross-sectional strength (members in braced frames and tapered members only),

b. overall member strength, and

c. lateral-torsional buckling strength.

Section Values of \( U_1 \)

In lieu of a more detailed analysis, the value of \( U_1 \) for the axis under consideration, accounting for the second-order effects due to the deformation of a member between its ends, shall be taken as:

\[ U_1 = \frac{\omega_1}{1 - C_u / C_e} \]

where

\[ \omega_1 = \text{ for the axis under consideration as defined in 13.8.4} \]

\[ C_e = \frac{\pi^2 EI}{L^2} \text{ for the axis under consideration} \]

D14.C.6.8 Combined Axial Tension and Bending

Members required to resist both bending moments and an axial tensile force shall be proportioned such that:

a. \[ \frac{T_u}{T_r} + \frac{M_u}{M_r} \leq 1.0 \]

where

\[ M_r = \phi M_p \text{ for class 1 and class 2 sections} \]

\[ = \phi M_s \text{ for class 3 and class 4 sections} \]

b. \[ \frac{M_u}{M_r} - \frac{T_u Z_{pl}}{M_r A} \leq 1.0 \text{ for class 1 and class 2 sections,} \]

\[ \frac{M_u}{M_r} - \frac{T_u Z_e}{M_r A} \leq 1.0 \text{ for class 3 and class 4 sections} \]

where

\[ M_r = \text{ as defined in 13.5 or 13.6} \]

D14.C.7 Design Parameters

The design parameters outlined in table below may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.
The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

**Note:** Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

### Table 211: South African Steel Design Parameters

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEAM</td>
<td>0</td>
<td>0 - Perform design at ends and those locations specified in the section command. 1 - Perform design at ends and 1/12th section locations along member length.</td>
</tr>
<tr>
<td>CAN</td>
<td>0</td>
<td>Deflection check for cantilever members. 0 - False 1 - True</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Greater than 0.0 and less than 2.5, Value of $\omega_2$ (C1.13.6) to be used for calculation  Equal to 0.0: Calculate $\omega_2$</td>
</tr>
<tr>
<td>CMY</td>
<td>1.0</td>
<td>Values between 0.4 – 1 used for $\omega_{1y}$ (C1.13.8.5). Input beyond the values consider as 1.</td>
</tr>
<tr>
<td>CMZ</td>
<td>1.0</td>
<td>Values between 0.4 – 1 used for $\omega_{1z}$ (C1.13.8.5). Input beyond the values consider as 1.</td>
</tr>
<tr>
<td>DFF</td>
<td>0</td>
<td>Default is 0 indicating that deflection check is not performed</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DJ1</td>
<td>0</td>
<td>Start node of physical member for determining deflected pattern for deflection check and should be set along with DFF parameter</td>
</tr>
<tr>
<td>DJ2</td>
<td>0</td>
<td>End node of physical member for determining deflected pattern for deflection check and should be set along with DFF parameter</td>
</tr>
<tr>
<td>DMAX</td>
<td>1,000</td>
<td>Maximum allowable depth</td>
</tr>
<tr>
<td>DMIN</td>
<td>0</td>
<td>Minimum required depth</td>
</tr>
<tr>
<td>FU</td>
<td>345 Mpa</td>
<td>Ultimate strength of steel</td>
</tr>
<tr>
<td>FYLD</td>
<td>300 Mpa</td>
<td>Yield strength of steel</td>
</tr>
<tr>
<td>KT</td>
<td>1.0</td>
<td>K value for flexural torsional buckling</td>
</tr>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local Y-axis, usually minor axis</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local Z-axis, usually major axis</td>
</tr>
<tr>
<td>LT</td>
<td>Member length</td>
<td>Length for flexural torsional buckling</td>
</tr>
<tr>
<td>LY</td>
<td>Member length</td>
<td>Length in local Y axis for slenderness value KL/r</td>
</tr>
<tr>
<td>LZ</td>
<td>Member length</td>
<td>Length in local Z axis for slenderness value KL/r</td>
</tr>
<tr>
<td>MAIN</td>
<td>0</td>
<td>Flag for controlling slenderness check</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members</td>
</tr>
</tbody>
</table>
**Parameter Name** | **Default Value** | **Description**
--- | --- | ---
**RATIO** | 1.0 | Permissible ratio of applied load to section capacity
| | | Used in altering the RHS of critical interaction equations

**SSY** | 0 | Sidesway parameter
| | | 0 - Sideway about local Y-axis.
| | | 1 - No sideway about local Y-axis.

**SSZ** | 0 | Sidesway parameter
| | | 0 - Sideway about local Z-axis.
| | | 1 - No sideway about local Z-axis.

**STIFF** | Member length | Center-to-center distance between transverse web stiffeners.

**TRACK** | 0 | Track parameter
| | | 0. Print the design output at the minimum detail level.
| | | 1. Print the design output at the intermediate detail level.
| | | 2. Print the design output at maximum detail level.

**UNB** | Member Length | Unsupported length in bending compression of bottom flange for calculating moment resistance

**UNT** | Member Length | Unsupported length in bending compression of top flange for calculating moment resistance

---

**D14.C.8 Code Checking and Member Selection**

**D14.C.8.1 Code Checking**

The purpose of code checking is to determine whether the current section properties of the members are adequate to carry the forces obtained from the most recent analysis. The adequacy is checked as per the SAB0162-1: 1993 requirements.

Code checking is done using forces and moments at specified sections of the members. If the BEAM parameter for a member is set to 1 (which is also its default value), moments are calculated at every twelfth point along the
beam. When no section locations are specified and the BEAM parameter is set to zero, design will be based on member start and end forces only. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. Using the TRACK parameter can control the extent of detail of the output.

Refer to D1.B.1.3 Code Checking (on page 1418) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2852) for details the specification of the Code Checking command.

**D14.C.8.2 Member Selection**

The member selection process involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially.

For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on members listed as PRISMATIC.

Refer to D1.B.1.4 Member Selection (on page 1419) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2853) for details the specification of the Member Selection command.
This section describes how to review output, perform post-processing tasks, generate reports, and plot from STAAD.Pro.

P. To view analysis results

There are several ways to open analysis results in the STAAD.Pro Editor.

1. Either:
   - On the **Utilities** ribbon tab, select the **View Analysis Output** tool in the **Utilities** group.
   - or
   - Drag a STAAD output file (file extension .anl) icon from Windows Explorer and drop it on to the STAAD.Pro Editor icon.
   - or
   - In the Windows Explorer, double-click the icon for the file, Bentley.Staad.Editor.exe.

2. If you opened the STAAD.Pro Editor, then:
   - a. select **File > Open**.
   - b. Select **STAAD Analysis Output Files (*.anl)** in the file type drop-down list.
   - c. Navigate to and select a STAAD output file and the click **Open**.

   The output results open in the STAAD.Pro Editor application.

**Related Links**
- [I. STAAD.Pro Editor](on page 2251)
P. Postprocessing Workflow

This workflow offers graphical result verification and visualization facilities. A comprehensive custom report
generation facility is also incorporated. The customized reports may contain tabular results as well as graphics.

P. To reposition support reaction labels

To reposition support reaction labels so they do not overlap, use the following procedure.

1. Select the View window to ensure it has focus.
2. Select the Select Text tool in the Selection group on the Results ribbon tab.
   The mouse pointer changes to the text selection cursor.
3. Click on any annotation and drag (while holding down the left mouse button) it to the desired location in the
   View window.
   The leading arrow is redrawn to the new annotation position.
4. Repeat step 3 with other annotations you wish to reposition.
5. Press <Esc> to return to the default selection mode.

P. To create a animated video file from analysis results

AVI files are a mechanism by which a dynamic result, such as, a deflection diagram in animation, may be
captured and recorded. In an animated view, the movement from one extremity to the other is captured as
several frames. Presently, this facility is available in STAAD for deflection, section displacement, mode shape and
plate stress contour diagrams.

1. Perform a successful analysis on a STAAD input file.
2. On the Utilities ribbon tab, select the Create AVI File tool in the Utilities group.
   The Create AVI File dialog opens.
3. (Optional) Change the values of the Total No. of Frames and Frame Rate /sec to alter the speed and length
   of the animation.
4. Select a result to animate: Deflection, Sectional Displacement, Mode Shape, or Stress Contour.
   Note: Mode Shape and Stress Contour are not active if the required data of that type are not present in the
   STAAD file, such as a modal extraction, or finite elements.
5. Click OK.
   A Save As dialog opens.
6. Click Save.
   The Video Compression dialog opens.
7. (Optional) Select the means of Video Compression (codec) to use and change settings for the selection as
   needed.
8. Click OK.
   A message indicating the status of the AVI creation opens.
9. Click OK.
   The AVI Video window opens and begins playing the saved animation.
P. Nodal Results

This section describes how to display and review analysis results for nodes.

P. To display reactions at each support

1. Select the Reactions page in the Postprocessing workflow. The reactions are labeled for the current load.
2. On the Results ribbon tab, select the Annotate tool in the Configuration group.

   The Annotation dialog opens.
3. Select the Reactions tab.
4. Select the Show Line option.
5. (Optional) Change the scale for force and moment if needed.
6. Click Annotate to display the reactions as scaled load arrows.
7. Click Close.

Related Links
- Annotation dialog (on page 3145)
- Annotation dialog (on page 3145)

P. Beam Results

This section describes how to display and review analysis results for members.

P. To display results diagrams on the members

To display the results diagrams drawn on to the members, use the following procedure.

1. On the Results ribbon tab, select one of the graphical results tools in the View Results group:
Tip: Alternatively, you can select the **Layout > Beam Forces** tool in the **View Results** group on the **Results** ribbon tab to display the major axis bending graphs on the members (MZ).

The selected diagram is drawn on the members.

**Note:** The selected tool is shaded to indicate it is active.

2. (Optional) Either:
   - repeat step 1 to select different diagrams as needed
   - select the active diagram tool to hide the diagram

   **Tip:** Multiple diagrams may be displayed simultaneously.

3. To change the scale of any diagrams:
   a. On the **Results** ribbon tab, select the **Scale** tool in the **Configuration** group.

   The **Diagrams** dialog opens to the **Scales** tab.

   b. Change the scale factor for the relevant item in the **Results Scales** group.

   **Tip:** Smaller values result in more exaggerated diagram sizes.

   c. Click **Apply** to update the diagram.

   d. Click **OK** to close the dialog.

**P.** To view stress contour of a beam

To display the full section stress contour on a beam

Stress contours can be displayed for sections that have predefined dimensions (e.g., catalog and prismatic sections) and general sections with stress disances defined.

1. On the **Results** ribbon tab, select the **Layouts > Beam Stress** tool in the **View Results** tab.
   The **3D Beam Stress Contour** dialog opens.

2. In the **Whole Structure** view window, click on any member to select it.

   The **Select Section Plane** dialog opens.

   The **3D Beam Stress Contour** displays a rendered view of the member with the stress contours on the surface on the left and the current cross-section stress contour on the right. Compression is display in blue and tension in red.

   The table displays the corner stress values at points along the member length.
3. Use the slider in the Select Section Plane dialogs to update the cross section stress contour display.

4. (Optional) In the Select Section Plane dialog, type a Y Point and Z Point value to display the cross-section stress at an arbitrary point within the section.

P. To display steel design utilization ratios

To display member utilization values and color-coded design results on the structure, use the following procedure.

The utilization ratios for members are only available for steel design performed in the batch mode.

The utilization ratio is for the load envelope used for design and is not affected by the current load case in the Postprocessing workflow.

1. On the Results ribbon tab, select the Layouts > Utilization tool in the View Results group.
   The utilization ratio label is shown next to each member included in the steel design.

   Note: Members are color coded: green for passing design, blue for failing less than 50% over utilization, and red for failing more than 50% over utilization.

   The Design Results table includes a tab for All members as well as for Failed members.

2. (Optional) Change the color coded limits:
   a. On the Results ribbon tab, select the Structure tool in the Configuration group.
      The Diagrams dialog opens.
   b. Select the Design Results tab.
   c. In the color-coding table, change the To cell values for the green and blue rows.
      For example, you may want to change the green (passing) utilization ratio to 1.003 and the blue (nearly passing) utilization ratio to 1.05 to get a “feel” for where the members are nearly adequate.
   d. Click Apply.
      The diagram color coding updates accordingly.
   e. Click OK.

P. To display bending and shear diagrams

To display the bending (both major and minor axes) and shear diagrams, use the following procedure.

1. On the Results ribbon tab, select the Layouts > Graphs tool in the View Results group.
   The Beam Graphs windows open.

2. Select a member in the view window.

3. Select the current Load on the Results ribbon tab in the View Results group.

4. Change the diagrams displayed in any of the Beam Graphs windows:
   a. Right-click and select Diagrams from the pop-up menu.
      The Diagram dialog opens.
   b. Set the checkbox for the graphs you want to add to the window.
   c. Click OK.
      The diagrams are updated.

P. Plate Results

This section describes how to display and review analysis results for plate elements.
P. To display plate results contours

To display analysis results of plate elements as contour lines on the plates, use the following procedure.

Plate stresses can be displayed as contour lines along the plate elements graphically for a variety of stress types.

1. Either:
   - on the Results ribbon tab, select the Plate Stress tool in the View Results group
   - or
   - select the Plate Results page in the Postprocessing workflow bar
   
   The Diagrams dialog opens to the Plate Stress Contour tab.
   
   The tables in the Plate Results page also display results data for many of the stress types.

2. (Optional) Select the Load Case.
   
   Tip: The current load case in the Results ribbon tab will be selected.

3. Select the Stress Type to display.
   
   STAAD.Pro can display a variety of component element stresses or stress combinations.

4. Select the display options for the contours.

5. Click Apply.

6. Make any different selections to change the contour display as needed and then click Apply again to update the view.

7. Click OK.
   
   The dialog closes.

Related Links

- Diagrams dialog (on page 2927)

P. To display plate results along a cut line

To view the plate stresses along an arbitrary line cut across plate elements, use the following procedure.

1. On the Results ribbon tab, select the Layouts > Results Along Line tool in the View Results group.
   
   The Results Along Line dialog and layout open.

P. Dynamic Results
P. To display floor spectrum results

To display the generated floor spectra results for a time history load case, use the following procedure. These results are only available for a STAAD.Pro model with a time history load and a generated floor spectrum command.

1. On the **Results** ribbon tab, select the **Layouts > Floor Spectrum** tool in the **Dynamics** group.
   The **Floor Spectrum Graph** window and **Floor Spectrums** table open.

   **Tip:** The peak acceleration value is labeled.

2. On the **Floor Spectrums** table, select the **Floor Spectrum Set** from the drop-down list.
   The tabulated values and the graph update to display the selected set.

3. (Optional) You can change the display of the graphed data by right-clicking on the graph and selecting:
   - **Linear Graph** display the response spectrum graph with linear axes
   - **Log-Log Graph** display the response spectrum graph with logarithmic axes (default display)
   - **Show Points** display the individual calculated points on the plot

4. (Optional) To save the current set data to an external file:
   a. Right-click on the graph and then select **Save data in a text file** from the pop-up menu.
      A Windows **File Open** dialog opens.
   b. Select a location to save the data.
   c. Type a **File Name** and then click **Save**.
   The frequency and acceleration data is saved in a plan text file (file extension .txt).

5. You can use pictures of the graph in other reports or external programs by right-clicking on the graph and selecting:
   - **Copy Picture** copy the graph to the Windows clipboard
   - **Take Picture** save the image for use in reports for later use. A Picture dialog will open for a label and caption. Click **OK** to save the graph to the report Picture Album.

**Related Links**
- [AD.2007-04.1.2 Floor Response Spectrum](on page 199)

P. Reports

STAAD.Pro can generate the following specialized reports.

P. To generate a node displacement report

1. In the **Postprocessing** mode, use the **Node Cursor** tool to select nodes for inclusion in the report.
2. On the Results ribbon tab, select **Reports > Node Displacements** in the **Reports** group.
The Node Displacement dialog opens.
3. (Optional) Select the sorting criteria you want to use in the report.
4. (Optional) Select the Load Cases on the Loading tab you want to include in the report.
5. Specify the report metadata:
   a. Select the Report tab.
   b. Type the Title for the report.
   c. (Optional) Check the Save Report option and type an Id value if you want to change it.
6. Click OK.

P. To generate transfer forces report

To generate transfer forces for a selected set of beams and columns to include in your report, use the following procedure.

1. In the Postprocessing workflow, use the Beam Cursor tool to select all the members framing into a connection.
2. On the Results ribbon tab, select Reports > Column Transfer Force in the Reports group.

The Transfer Force for Selected Members dialog opens.
3. (Optional) Click any load cases you to exclude from the transfer forces report.
   By default, all load cases will be included.
4. Click Insert to Table.
   The Transfer Force Report table opens.
5. Click Close.

The Transfer Force Report can be found within the Reports group on the Items tab in the Report Setup dialog.
Related Links
- Transfer Forces for Selected Members dialog (on page 3150)

P. To generate a floor vibration report

To generate a floor vibration report per the procedures of Chapters 3 and 4 of the AISC Steel Design Guide Series No. 11, use the following procedure.

Floor vibration reports are performed for composite decks only. You must define and run an analysis for a composite deck structure in order to generate a floor vibration report.

1. On the Results ribbon tab, select the Reports > Floor Vibration Report tool in the Reports group.
The **Floor Vibration Report** dialog opens.

2. Select the composite deck name from the **Select** drop-down list and then select the **Load Case**.

3. Either:
   - select the closes description for the type of building in the **Bldg Type** drop-down list in order to select a predefined Beta value
   - or
   - type the **Beta** value directly

4. Click **Check**. The results of the vibration check are displayed.

5. **(Optional)** Click **Print** and then select the printer to which you want to send the copy.

   **Note:** The comparison check is the **Acceleration** versus the **Allowed Acceleration** in the Check for Combined Mode section.

   **Note:** Floor vibration reports must be output from this dialog. They cannot be included in the STAAD.Pro report.

**Related Links**

- *AD.2005.2.1 Floor Vibration Analysis* (on page 365)
- *P. Floor Vibrations Engineering Theory* (on page 2221)
- *Floor Vibration Output dialog* (on page 3150)
P. Floor Vibrations Engineering Theory

The fundamental natural frequency of the joist mode and the girder mode can be determined from equation 3.3 on page 11 of the design guide:

\[ f_{j/g} = 0.18 \left( \frac{g}{\Delta} \right)^{1/2} \]

where

\( f_{j/g} \) = fundamental natural frequency of the joist or the girder mode

\( g \) = acceleration due to gravity

\( \Delta \) = midspan deflection of the member due to the weight supported.

For the combined mode, the fundamental natural frequency can be determined from equation 3.4 on page 11 of the design guide:

\[ f_n = 0.18 \left[ \frac{g}{(\Delta_j + \Delta_g)} \right]^{1/2} \]

where

\( f_n \) = fundamental natural frequency for the combined mode

\( g \) = acceleration due to gravity

\( \Delta_j \) = joist deflection due to the weight supported

\( \Delta_g \) = girder deflection due to the weight supported

\( \Delta_j \) and \( \Delta_g \) are the local deflection of the joist and the girder determined from a secondary operation after the analysis. The stiffness analysis will yield the global deflection values for the girder beams. A line joining the start and the end nodes of the girder beam in its deflected position is created as a base line. Relative to this base line, the deflection values are zero for the start and end nodes. The local deflection values of the intermediate points of the girder beam are evaluated from the global deflection values relative to this base line.

It is this local deflection that is used in calculating the fundamental natural frequency as shown in the earlier equations. Further, the local deflection is also used in calculating the equivalent uniform loading on the joist and the girder, \( w_j \) and \( w_g \), as shown in the equation on page 21 of the AISC Design Guide:

\[ \text{where} \]

\( I_{j/g} \) = moment of inertia of the effective joist or girder section

\( L_{j/g} \) = joist or girder span

\( E_g \) = modulus of elasticity of steel

In addition to the terms \( f_j, \Delta_j, w_j \) shown above, the following additional terms — \( D_s, D_j, B_j, \) and \( W_j \) — which are explained below, are also reported for the joist mode.

where

\( D_s \) = the transformed slab moment of inertia per unit width and is determined from the equation at the bottom right corner of page 17 of the AISC Design Guide:

\( d_e \) = effective depth of the concrete slab, usually taken as the depth of the concrete above the form deck plus one-half the depth of the form deck

\( N \) = dynamic modular ratio = \( E_s / 1.35 E_c \)

\( E_c \) = modulus of elasticity of concrete

\( D_j \) = the joist or the or beam transformed moment of inertia per unit width, and is determined from the equation shown at the top left of page 18 of the AISC Design Guide:

\[ = \frac{I_j}{S} \]

\( S \) = joist or beam spacing
$B_j =$ the effective width for the beam or joist panel mode and is determined from equation 4.3a on page 17 of the AISC Design Guide:

$$= C_j \left( \frac{D_s}{D_j} \right)^{1/4} L_j \leq \frac{2}{3} \times \text{floor width}$$

$C_j =$ 1.0 for interior panels

2.0 for edge panels

$W_j =$ the weight of the beam panel and is calculated from equation 4.2 on page 17 of the AISC Design Guide and page 21 left side:

$$W_j = w_j B_j L_j \ (x \ 1.5 \ 	ext{if continuous})$$

For the girder mode, the terms reported include $f_g, \Delta_j, w_g$ which were explained earlier, and $D_g, B_g, W_g$ which are described below.

where

$D_g =$ the girder transformed moment of inertia per unit width described on page 18 of the AISC Design Guide:

$$= \frac{I_g}{L_g} \text{ for all but edge girders}$$

$$= \frac{I_g}{2L_g} \text{ for edge girders}$$

$B_g =$ the effective width for the girder panel mode and is determined by equation 4.3b on page 18:

$$= C_g \left( \frac{D_g}{D_j} \right)^{1/4} L_g \leq \frac{2}{3} \times \text{floor width} \text{ for interior panels}$$

$$= \frac{2}{3} L_g \text{ for edge panels}$$

$C_g =$ defined on page 18 as:

$$= 1.6 \text{ for girders supporting joists connected to girder flange (e.g., joist seats)}$$

$$= 1.8 \text{ for girders supporting beams connected to the girder web.}$$

$W_g =$ the weight of the girder panel and is calculated by equation 4.2 on page 17 and described on page 21:

$$= w_g B_g L_g \ (x \ 1.5 \ \text{if continuous})$$

For the combined mode of vibration the parameters reported are $f_n, W, \beta$, Peak Acceleration and Acceleration Limit.

$f_n$ is calculated from equation 2 shown above.

$W$ is the equivalent panel weight in the combined mode and is calculated from the equation shown on page 21 of the AISC Design Guide:

$$W$$

$\beta$ is the value of the damping ratio as per Table 4.1 on page 18 of AISC Design Guide.

The peak acceleration due to walking excitation is then determined from the equation 4.1 on page 17 and on page 21 of the AISC Design Guide:

where

$$\alpha_p =$ peak acceleration value due to walking excitation

$P_0 =$ a constant force representing excitation and is determined as per Table 4.1 of the Design Guide.

$f_n =$ fundamental natural frequency in combined mode

The acceleration limit is determined from Table 4.1 on page 18 of the Design Guide.
P. Postprocessing Workflow

Related Links

- Floor Vibration Output dialog (on page 3150)
- P. To generate a floor vibration report (on page 2219)

P. Reviewing Pushover Analysis Results

This section describes how to view some of the specific results from a pushover analysis.

Once a successful pushover analysis is performed, pushover results are available in the Postprocessing workflow.

P. To review pushover load steps

To review the pushover load applied at each load step, use the following procedure.

1. On the Results ribbon tab, select the Layouts tool drop-down in the Dynamics group.
2. Select the Loads tool in the Pushover group.
   The Load Value table opens.
3. Either type a integer or use the arrows in the Select Load Step field.
   The pushover loads at each node and base shear are displayed for this load step.

P. To view capacity curve and determine target displacement

To determine the performance point of a pushover analysis for a given performance level, use the following procedure.

The capacity curve displays the plot of the base shear versus the displacement at control joint.

1. On the Results ribbon tab, select the Layouts tool drop-down in the Dynamics group.
2. Select the Graphs tool in the Pushover group.
   The Capacity Curve graph (on page 3153) and table opens.
3. Right-click on the Capacity Curve graph and select Determine Target Displacement from the pop-up menu.
   The Define Modification Factor C0 dialog opens.
4. Specify the Modification Factor C0:
   a. Either:
      To...                        Do the following...
      use a table value           select the Select from Table option and then select a C0 factor value in the table
      use an interpolated value   select the Define option and then select or type the specific building parameters using the options
   
      The Modification Factor C0 value to be used is displayed in the dialog.
   b. Click Next.
      The Define Cm to Calculate Modification Factor C1 dialog opens.
5. Specify the Modification Factor C1:
   a. Select the Damping ratio (in percent) from the drop-down list.
   b. Select the effective mass factor, Cm, to use by clicking the value in the table.
Postprocessing and Reports
P. Postprocessing Workflow

The \( C_m \) value to be used is displayed in the dialog.

c. Click Next.

The Define Modification Factor \( C_2 \) dialog opens.

6. Specify the Modification Factor \( C_2 \):
   a. Select the Structure Performance Level from the drop-down list.
   b. Select the Framing Type from the drop-down list.
      The Modification Factor \( C_2 \) value to be used is displayed in the dialog.
   c. Click Finish.

The calculated target displacement and base shear at target displacement values are displayed on the Capacity Curve graph.

7. (Optional) Right-click on the Capacity Curve graph and select Show Idealized Capacity Curve from the pop-up menu.
   The idealized capacity curve is shown in blue on the graph, along with the curve parameters. These include the slopes of the linear portions of the idealized capacity curve along with the base shear and deflection at the inflection point.

8. (Optional) Repeat steps 3 through 6 and select a different Structure Performance Level as needed.

P. To display pushover node results

To display the nodal displacements and support reactions for a pushover analysis, use the following procedure.

1. On the Results ribbon tab, select the Layouts tool drop-down in the Dynamics group.
2. Select the Node Results tool in the Pushover group.
   The Node Displacements and Support Reactions tables open.
3. Either type a integer or use the arrows in the Select Load Step field.
   The tables display the results for this load step. The view window displays the deflected shape of the structure for this load step.

P. To review pushover beam results

Once a successful pushover analysis is performed, pushover results are available in the Postprocessing workflow.

1. On the Results ribbon tab, select the Layouts tool drop-down in the Dynamics group.
2. Select the Loads tool in the Pushover group.
   The Beam Hinge Results and Beam Force Detail tables open.
3. Either type a integer or use the arrows in the Select Load Step field.
   The values in the tables are updated for this load step. The hinges are graphically displayed on the structure for each load step using the color coding for acceptance criteria:

   - (Green) Initial Hinge format at the top of the elastic range, represents Immediate Occupancy.
   - (Blue) Hinge in the Life Safety range
   - (Purple) Hinge in the Collapse Prevention range.
   - (Red) Hinge greater than Collapse Prevention range.

Related Links
Postprocessing and Reports

P. Postprocessing Workflow

- \textit{G.17.4.1.6 Frame element hinge properties} (on page 2382)
- \textit{Capacity Curve graph} (on page 3153)

P. Pages in the Postprocessing Workflow

- Displacements
- Reactions
- Beam Results
- Plate Results
- Solid Results
- Dynamics
- Reports

P. \textbf{Node Displacements} table

Displays the displacements and rotations for each node for the selected load case. For geometric nonlinear analysis, you can select the load step for which you want to display the displacements.

Opens when the \textit{Displacements} page is selected in the \textit{Postprocessing} workflow.

\begin{itemize}
  \item \textbf{Select Load Step} \quad Select the Load Step value to display in the view window, \textit{Node Displacement Curve} dialog, and the \textit{Node Displacements} table. The Maximum Number of Load Steps for the current load case is displayed on the right-hand side of the table controls.
  \item \textbf{Limit Maximum Load Step for Graph} \quad Select this option to limit the Load Level scale on the Node Displacement Curve to the Selected Load Step.
\end{itemize}

P. \textbf{Support Reactions} table

Used to display values of support reactions in tabular format. Three tabs help organize the results: All, Summary, and Envelope.

Opens when the \textit{Reactions} page is selected in the \textit{Postprocessing} workflow.

\begin{itemize}
  \item \textbf{All tab} \quad Displays the reactions at support nodes for all load cases.
  \item \textbf{Summary tab} \quad Displays the maximum and minimum support reactions (translational and rotational) for each degree of freedom. All nodes and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).
  \item \textbf{Envelope tab} \quad Displays maximum and minimum support reaction envelopes along with the Load Case number that caused the reaction. All degrees of freedom and all specified Load Cases are considered.
\end{itemize}
P. Generating Reports

P. To setup report contents

To select the items to include in a STAAD.Pro report, use the following procedure.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Select the **Report** tab and then click **Setup**.
   The **Report Setup** dialog opens.
3. Select the **Items** tab and then specify the sections to include in the report.
   The list on the right contains the included report items in the order they will appear in the report.
   a. Select a report category in the drop-down list.
      The list of available items updates.
   b. Either:
      Select one or more items in the available list and then click > to add to the report.
      **Tip**: Hold <Ctrl> to click on multiple items. Hold <Shift> to select a series of adjacent items.
      or
      Click >> to include all items within the report category in the report.
   c. Repeat steps 3a and 3b to include items from additional report categories.
4. (Optional) Use the up and down buttons to reorder items in the report list.
5. (Optional) Click **Print** to output the report immediately.
   The Print dialog opens to select the system printer.
6. Click **OK**.

You can limit the report contents by load cases, modes (for dynamic results), and ranges of objects.

P. To add a custom header and logo to reports

To add custom header text and a logo for your organization to your reports, use the following procedure.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Select the **Report** tab and then click **Setup**.
   The **Report Setup** dialog opens.
3. Select the **Name and Logo** tab.
4. In the text field, type the custom header text.
5. (Optional) Customize the text placement and appearance:
   a. Click **Font** to select a font face, style, and size.
   b. Select the font **Alignment**.
      This is active when the logo is aligned to right or left.
6. Click **File** to locate and select a bitmap image (file extension `.bmp`) to use for your header logo.

### P. Steel AutoDrafter Workflow

The Steel AutoDrafter workflow extracts planar drawings and material take-off from a structural steel model prepared in STAAD.Pro.

Steel AutoDrafter produces plans at any level and sections in any of the orthogonal directions.

**Note:** The model needs to be analyzed and saved before using Steel AutoDrafter workflow.

#### Overview

![Steel AutoDrafter Workflow](image)

**Related Links**

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 94)

#### P. To open the Steel AutoDrafter workflow

You must perform an analysis before using the Steel AutoDrafter workflow.

1. Perform a successful analysis.
2. Select **Steel AutoDrafter** in the **Workflows** panel.
The **Steel AutoDrafter** ribbon tab and the Steel AutoDrafter workflow **Layout** page open.

**Related Links**
- [Steel AutoDrafter tab](on page 3157)

**P. To configure units in Steel AutoDrafter**

To change the system of units for the interface as well as in drawings and material take-off, use the following procedure.

1. On the **Steel AutoDrafter** ribbon tab, select the **Drawing Style Manager** tool in the **Edit** group.

   ![Drawing Style Manager](image)

   The **Drawing Style Manager** dialog opens.

2. Select the **Appearance** tab.

3. Select the **Drawing Unit** to use.

   Metric
   or
   English

4. Click **OK**.

   Grids, material take-offs, etc. are all now provided in the selected system of units.

**Related Links**
- [Drawing Style Manager dialog](on page 3160)
- [Member Labels](on page 3161)

**P. Drawing**

To create a drawing, select any element and select the type of drawing to be generated using the options provided in the menu.

The options provided in the menu are dynamic based on the type of member.

Based on the member selected graphically. The context menu shows the possible drawings that can be generated.

The drawing to be generated can be on demand or can be added to the list of drawing.

For Projected View, Options are provided in drawing group tab.

   e.g.

   For a Horizontal member, Options to draw plan (XZ Plane) and elevation in the plane of the member are displayed.

   For a Vertical member, Options to draw elevations in XY or YZ planes are displayed.
Drawing Settings

Drawing can be generated with either using full section or part section.

Full Section: All edges of the section are drawn for the full length of the member

Part Section: All edges of the section are drawn for the part length of the member

The Scale of drawing can be set as 1:25, 1:50, 1:100.

You can draw the section directly or add it to the drawing list.
The TOS (Top of Steel) for beams is adjusted as Y coordinate of the floor. Accordingly the center line of the beam is shifted down by half the beam depth.

For Built-up section made of two members, batten plates are drawn as indicative.
For Connection between concrete and steel, indicative base plates are shown.

For elevation Bracing's the angle is adjusted as per the change of center line of the beams as per TOS.

For Bottom Chord of the Trusses the node between bottom chord and the top chord is adjusted as per the size of the bottom chord member. Accordingly the inclination of the top chord is adjusted without changing the top of truss. Similarly the lengths of the members connecting bottom chord and top chord are adjusted.

Similar to Truss, for the inclined portals an indicative joint with connection plates is shown between the column and the portal member.

For Moment connection there is no gap provided between the connecting members. For other connections an indicative gap is shown between the connecting members.

Notes

Drawing generated are as per the members on a given plane, Any members that cut across different planes are not shown.

Curved members are currently not supported.

Members with varying cross sections are currently not supported.

In case if the section defined in STAAD.Pro is not identified by Steel AutoDrafter, Then the member will not be displayed in drawing.

P. To edit grid labels

To toggle the display of grids and elevation marks or to change the label for each, use the following procedure.

Steel AutoDrafter identifies the grids from column (vertical elements) centers from the analysis file.

**Note:** The model in STAAD.Pro is required to be created using Y up axes system.

**Tip:** Grid can be imported from another adfx file using the import grid option provided. The adfx file will be available along with the relevant STAAD.Pro file in the same folder. Click Import to select a adfx file.

1. On the **Steel AutoDrafter** ribbon tab, select the **Grids** tool in the **Edit** group.

   ![Grid Manager dialog](image)

   The **Grid Manager** dialog opens.

2. To toggle the display of a grid line or elevation mark, use the check box associated with that grid or elevation mark.

3. To edit the label of a grid or elevation mark, type the label in **Mark** cell associated with that grid or elevation mark.

4. Click **OK**.

**Related Links**

- Grid Manager dialog (on page 3159)
P. Built-Up Sections in Steel AutoDrafter

Steel AutoDrafter recognises and draws built-up sections. These sections have to be defined in STAAD.Pro, using a particular format within the user table section.

<table>
<thead>
<tr>
<th>Section</th>
<th>User Table Prop Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle Star</td>
<td>AS-ISA75X75X8-10</td>
</tr>
<tr>
<td>I/H Star</td>
<td>IS-ISMB500</td>
</tr>
<tr>
<td>Built Up I</td>
<td>IBU-DPT-WD-WT-FT</td>
</tr>
<tr>
<td>Built Up Channel</td>
<td>CBU-DPT-WD-WT-FT</td>
</tr>
<tr>
<td>Two I/H Sxn</td>
<td>2I-ISMB250-100</td>
</tr>
</tbody>
</table>

where

\[
\begin{align*}
DPT &= \text{total depth of the section} \\
WD &= \text{total width of section} \\
WT &= \text{web thickness} \\
FT &= \text{flange thickness}
\end{align*}
\]

“10” in the Angle Star represents the clear gap between the two angles.
“100” in the 2I represents the clear gap between the two sections.

Example of creating user table in STAAD as per above format:

```
START USER TABLE
TABLE 1
UNIT CM MTON
GENERAL
IEU-750-250-12-20
185.2 75 1.2 25 2 169049 5218.56 85.28 4507.98 417.48 0 0 0 0 0 0
IS-ISMB200
61.6 20 0.57 10 1 2257 14 225.7 451.4 0 0 0 0 0 0
AS-ISA75X75X6-8
17.32 15 0.58 15 0.58 192.58 148.6 1.97 33.62 28.02 12.99 12.99 51.47 -
44.04 19690.9 13.84
2I-ISMB200-200
64.66 20 1.62 30 1.07 6766 4470.8 21.85 451.067 447 48.49 48.49 656.48 -
517.95 218592 16.76
END
```

P. Drawing List & Groups panel

Drawings that are to be created can be added to the Drawing List using any of the following options. Groups of members in drawings can be managed here.
Drawing List tab

**Section Description**

This is used to add drawings to the list.

- By selecting a member in the view area and then selecting the required drawing from the options in the Drawings List tab.
- By using the options in the Section Description.
- A projected view can be created by selecting three nodes.

Drawings added to the list can all be generated at once using the draw options

New Option in the list can be used to generate drawings together in a single sheet. (Default each drawing is generated in a new sheet)

Selection of points for projected view
P. To draw a plan

To create a plan drawing of members, use the following procedure.

1. Either:
   - right-click on a horizontal member and then select **Draw Plan @Y = elevation** from the pop-up menu
   - or
   - In the **Drawing List** tab, select **Plan** from the **Type** drop-down list
     - If you used the right-click pop-up menu, the **Drawing Settings** dialog opens.

2. Select the **Scale** for your drawing from the drop-down list.

3. Select either **Full Sections** or **Part Sections** from the **Draw As** drop-down list.
   - This will control the extents of how members are drawn.

4. If you are using the **Drawing List** tab, then select the **Location** to use from the drop-down list.
   - If you used the right-click pop-up menu, then the grid line of that member is automatically used.

5. Either:
   - click **Draw** to generate the drawing
   - or
   - click **Add to Drawing List** to add this drawing to the Drawing List tab

If you selected to add this drawing to the drawing list, then click the **Draw** tool below the list on the **Drawing List** tab to generate all of your drawings.
To open a drawing in MicroStation

To open the current drawing in MicroStation, use the following procedure.
You must have either MicroStation or Bentley PowerDraft installed to use this feature.

**Note:** If you have MicroStation PowerDraft installed instead of MicroStation, you will have that product’s icon instead of the MicroStation icon in the drawing toolbar.

1. Open a drawing in the Steel AutoDrafter workflow.
2. On the drawing toolbar, select the **Open in MicroStation** tool.

You are prompted to save the drawing file (.dxf file extension).

The drawing opens in MicroStation.

**Tip:** The resulting .dxf can be opened within any compatible program.

P. To create a drawing group

To create groups (physical members) for inclined members such as bracing, truss chords, portals, etc., use the following procedure.

Steel AutoDrafter will identify physical members automatically for vertical and horizontal members.

1. Right-click on a member and select **Cut Section @<grid or plane>**.
The Section View dialog opens.

2. Select the group type tool corresponding to the group of members you want to create:

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top chord -</td>
<td>can include purlins</td>
</tr>
<tr>
<td>Bottom chord</td>
<td></td>
</tr>
<tr>
<td>Portal</td>
<td>can include purlins at a specified spacing</td>
</tr>
<tr>
<td>Bracing</td>
<td></td>
</tr>
<tr>
<td>Inclined Column</td>
<td></td>
</tr>
</tbody>
</table>

The mouse pointer changes to a selection box.

3. Select the members to be grouped graphically:
   - click on them individually
   - or
   - click-and-drag a window box around the members

**Tip:** Use the view control tools to assist in selecting members.

Members in the selection set are highlighted in red.
4. Right-click anywhere in the Section View dialog view area and then select Create Group from the pop-up menu.

For Portal and Top Chord groups, the Create Group for <group type> dialog opens. For other groups, the group is created.

5. (Optional) For Portal and Top Chord groups, select the Purlin Section shape and section size to use and then click OK.

Note: Purlins are not part of the analysis but this allows you to add these to the drawings for accuracy.

6. Repeat Steps 2 through 5 to create additional groups.

7. Click the Draw tool.

The Drawing Settings dialog opens.

8. Select the Scale for your drawing from the drop-down list.

9. Select either Full Sections or Part Sections from the Draw As drop-down list.

This will control the extents of how members are drawn.

10. Either:

   click Draw to generate the drawing
   
or
   click Add to Drawing List to add this drawing to the Drawing List tab

Tip: To delete a group, right-click on the group name in either the Section View dialog or in the Drawing Groups tab of the Drawing Lists & Groups panel and then select Delete from the pop-up menu.

Related Links

• Section View dialog (on page 2237)

Section View dialog

Used to create groups (physical members) for inclined members in a section.

Opens when Cut Section @grid or plane is selected from the right-click pop-up menu of a selected member in the view area.

A list of group at the current section selection is displayed on the left side tree.

Tip: A list of all groups in the model can be viewed on the Drawing Groups tab of the Drawing Lists & Groups panel.

The view area of the dialog displays the members in the current section. You can manipulate this section view using the view controls to the right.
Postprocessing and Reports
P. Steel AutoDrafter Workflow

View Area Right-click Pop-up Menu

**Create Group**
(available when you have selected members to place in a group type) For bottom chord, bracing, and inclined column group types, the group is created.
For top chord or portal group types, the **Create Group for <group type>** dialog opens to specify purlin parameters.

**Show Node Number**
Toggles the display of node numbers in the dialog view area.

**Show Member Number**
Toggles the display of member numbers in the dialog view area.

**Show Member Property**
Toggles the display of member section sizes in the dialog view area.

- Top chord - can include purlins
- Bottom chord
- Portal - can include purlins at a specified spacing
- Bracing
- Inclined Column

Opens the **Drawing Settings** dialog, which is used to select the drawing scale, member extents drawn, and if the drawing is to generated immediately or saved to the drawing list.

Related Links
• *P. To create a drawing group* (on page 2235)

P. To generate a material take-off

1. Steel AutoDrafter generates material take-offs for the entire structure in two formats:
   - **Format**
   - **Select...**
   - **drawing** the **Drawing** tool in the **Material Take Off** group on the **Steel AutoDrafter** ribbon tab
Format: Select... text

Select the Text tool in the Material Take Off group on the Steel AutoDrafter ribbon tab.

The material take-off opens in the selected format. The drawing format will open in the Drawing page. The text format will open in the Material Take Off page.

2. (Optional) Click the Save tool in the page toolbar to save the material take-off to an external file. The drawing is saved as a .dxf file and the text is saved as an .html file.

![Figure 200: Drawing material take-off](image)
P. Earthquake workflow

This STAAD.Pro workflow is used to check if the structure conforms to the basic geometric recommendations made in Eurocode 8 (EC8). This workflow is in addition to the normal post-processing workflow which gives the various analysis results. These checks are intended to give you a “feel” for the structure and are not mandatory to proceed to the design phase.

**Note:** This feature requires STAAD.Pro V8i release 20.07.05 or higher.

Eurocode 8: Part 1 [EN 1998-1-1:2004] contains specific requirements and recommendations for building structures that are to be constructed in seismic regions. Essentially, these fundamental requirements have been provided to ensure that the structures can sustain the seismic loads without collapse and also –where required– avoid suffering unacceptable damage and can continue to function after an exposure to a seismic event.

**Tip:** The program feature assumes Y as vertical and that seismic forces act in the global X and Z directions.

P. Using the Earthquake Workflow

To perform the geometry checks per Eurocode 8, STAAD.Pro must first calculate the center of mass for each defined floor. Specifying a response spectrum loading accomplishes this.

All the analysis methods for design and evaluation of the performance of a structure mentioned in Eurocode, as listed in the following, are all available in STAAD.Pro:

- Linear modal response spectrum analysis (i.e., a linear elastic analysis used in conjunction with an EC8 response spectrum load)
- Nonlinear static analysis (i.e., a pushover analysis)
Nonlinear dynamic analysis (i.e., a time-history analysis)

Note: In order to make use of the EC8 response spectrum load, a linear modal response spectrum analysis should be used. This is required to locate the center of mass which is needed to perform the associated geometry checks.

Loads are distributed by element stiffness to all members during analysis. Thus, for the purposes of Eurocode 8, all members are assumed to be “primary seismic” elements.

Note: The EC8 response spectrum load must be specified in the model in order to perform any further checks, design, or detailing per Eurocode 8.

Related Links
- D. To perform seismic design and detailing per EC8 (on page 1018)
- M. To add an EC8 response spectrum (on page 849)
- D. To Perform a Preliminary Design per EC2 in RC Designer (on page 1018)
- D. To perform seismic design and detailing per EC8 (on page 1018)

P. To open the Earthquake workflow

You must perform an analysis on a model with a load cases including an EC8 response spectrum load.

1. Perform a successful analysis.
2. Select Earthquake in the Workflows panel.

The EC8 Stiffness page opens along with the Floors dialog, Storey Stiffness table, and Soft Storeys table.

Tip: You can nominally use the pages in the Earthquake workflow in any order. However, it is recommended that you work through the checks in a left-to-right order (i.e., EC8 Stiffness, EC8 Plans, and then EC8 Elevations).

P. To check stiffness of a structure per EC8

To review the stiffness of a structure per Eurocode 8 requirements, use the following procedure.

This set of checks is to enables you to judge whether the stiffness characteristics of the structure meet the recommendations set by EC 8. This implementation will:

i. Display the center of mass and center of stiffness graphically for each floor to inform you of some inherent eccentricities and their variation along the height of the building.

ii. Calculate the total stiffness of every story in each direction and provide you with a ratio of the stiffnesses in the two directions. This aides you to judge whether the structure conforms to the condition in EC8 that the stiffness should be similar in both directions.

iii. Check to see if there are any soft stories in the structure and will highlight them in the GUI. A soft story is defined as a story that has a stiffness in a particular direction that is less than 70% of the stiffness of the story above.
Postprocessing and Reports

P. Earthquake workflow

Note: An example demonstrating a soft story is included as C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Eur\Eurocode8_TestGeo_stiff.std.

1. Select the **EC8 Stiffness** page in the **Earthquake** page control bar.
   The center of mass and center of stiffness are drawn at each floor level.
2. For a particular floor, you can expand the entry in the **Floors** dialog to review the location of these items.
   The selected floor and centers of mass or stiffness are highlighted in the view area.
3. The stiffness ratios are reported in the Storey Stiffness table for each floor as well as for the total structure.
4. Any soft stories detected by the program are listed in the Soft Storey table.

Note: The stiffness of only the vertical elements are considered while working out the total stiffness of a story in a particular direction as the seismic loads are primarily resisted by the vertical elements in a structure.

P. To check for plan irregularities

To check for plan irregularities per Eurocode 8 such as re-entrant corners, slenderness, and torsional radius, use the following procedure.

This set of checks is to enable you to be able to classify the building as being 'regular in plan.' Checks will be performed and the results will be displayed graphically in the GUI. The program primarily checks for the main three plan regularity conditions set out in EC-8 viz.

i. Checks for re-entrant corners in floor slabs – The program performs checks on each floor slab and reports whether the condition set out in EC-8 for re-entrant corners has been satisfied.
ii. Check for slenderness of structure – The program calculates a "slenderness ratio" for the structure based on the maximum dimensions on plan and reports whether the EC-8 criteria are satisfied.
iii. Checks for torsional radius: - The program calculates the torsional radius for each floor and checks against the conditions set out in EC-8.

Note: An example demonstrating a soft story is included as C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Eur\Eurocode8_TestGeo_plan.std.

1. Select the **EC8 Plans** page in the **Earthquake** page control bar.
   The **Check Regularity in Plan** dialog opens along with the **Slab Re-Entrant Corners** and **Torsional Radius Check** tables.

   **Tip:** You may have to re-arrange the dialog and tables to display all windows.

2. To check for re-entrant corners:
   a. In the **Check Regularity in Plan** dialog, expand Clause 4.2.3.2(3) - [Re-Entrant Corners].
   A child entry is present for each floor in the structure.
   b. Select each floor to review any detected re-entrant corners.
      The details of any detected re-entrant corners are displayed in the **Slab Re-Entrant Corners** table.
3. To check for structure slenderness, expand Clause 4.2.3.2(5) - [Slenderness] in the **Check Regularity in Plan** dialog.
   The slenderness ratio and status for the structure is reported.
4. To check torsional radius, select the **Torsional Radius Check** table.
   The ratio of each torsional radius check for each floor along with the status of the check is reported here.
To check for elevation irregularities

To check for elevation irregularities as defined by EC8, use the following procedure.

This set of checks enables you to classify the building as a 'regular in elevation' structure. Checks will be performed and the results will be displayed graphically.

**Note:** An example demonstrating a soft story is included as C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Eur\Eurocode8_TestGeo_elev.std.

1. Select the **EC8 Elevations** page in the **Earthquake** page control bar.
   The **Earthquake Elevation Criteria** dialog opens along with the **Elevation Regularity** table.
2. In the **Earthquake Elevation Criteria** dialog, select to check elevation along X-Axis or Z-Axis.
3. Select the appropriate elevation criteria which applies to your structure.
   **Note:** Refer to clause 4.2.3.3 and Figure 4.1 in Eurocode 8 (EN 1998-1:2004) for additional information.
4. Click **Perform Checks**.
   The calculated values and results of this check are displayed in the **Elevation Regularity** table.
5. Repeat steps 2 through 4 to check the other direction or for other criteria if needed.

**P. Pages in the Earthquake Workflow**

The Pages in the Earthquake workflow page control bar are described below in brief. Detailed description of these pages is available in the pages to follow.

<table>
<thead>
<tr>
<th>Earthquake Page</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EC8 Stiffness</td>
<td>Used evaluate whether the stiffness characteristics of the structure meet the recommendations set by EC 8.</td>
</tr>
<tr>
<td>EC8 Plan</td>
<td>Used to classify the building as being “regular in plan.” Checks will be performed and the results will be displayed graphically in the GUI. The program primarily checks for the main three plan regularity conditions set out in EC-8.</td>
</tr>
<tr>
<td>EC8 Elevations</td>
<td>Used to classify the building as a “regular in elevation” structure. Checks will be performed and the results will be displayed graphically.</td>
</tr>
</tbody>
</table>

**P. EC8 Stiffness page**

Used evaluate whether the stiffness characteristics of the structure meet the recommendations set by EC 8.

**Note:** The stiffness of only the vertical elements are considered while working out the total stiffness of a story in a particular direction as the seismic loads are primarily resisted by the vertical elements in a structure.

When the Eurocode 8 | Stiffness Checks page is selected, the **Floors** dialog, **Story Stiffness** table, and **Soft Story** table open. The active view window displays the Whole Structure with the Center of Mass and Center of Gravity plotted for each floor. This is used to inform you of some inherent eccentricities and their variation along the height of the building.
Postprocessing and Reports
P. Earthquake workflow

P. Floors dialog
Displays the Center of Mass (CM), Center of Stiffness (CS), and Eccentricity (difference between CM and CS) for each defined floor in the structure. This information is graphically plotted onto the structure diagram in the Whole Structure view window.

Opens when the Eurocode 8 | Stiffness Checks page is selected.

Tip: You can select one of the entries in the Floors dialog to have the corresponding item highlighted in the Whole Structure view window.

P. Story Stiffness table
The program calculates the total stiffness of every story in each direction and provides you with a ratio of the stiffnesses in the two directions. This aids you to judge whether the structure conforms to the condition in EC8 that the stiffness should be similar in both directions.

Opens when the Eurocode 8 | Stiffness Checks page is selected.

P. Soft Story table
The program will check if there are any soft stories in the structure and will display any detected soft story and the direction in this table. These stories will also be highlighted in the active View window. A soft story is defined as a story that has a stiffness in a particular direction that is less than 70% of the stiffness of the story above.

Opens when the Eurocode 8 | Stiffness Checks page is selected.

P. EC8 Plan page
Used to classify the building as being “regular in plan.” Checks will be performed and the results will be displayed graphically in the GUI. The program primarily checks for the main three plan regularity conditions set out in EC-8.

When the EC8 Plan page is selected, the Check Regularity in Plan dialog opens and the active view window displays the Whole Structure.

Selecting various structure elements listed in the Check Regularity in Plan dialog will open either the Torsional Radius Check table or the Slab Re-Entrant Corners table.

P. Check Regularity in Plan dialog
Displays the results of the following checks for the main three plan regularity conditions set out in EC-8:

i. Checks for re-entrant corners in floor slabs
   The program performs checks on each floor slab and reports whether the condition set out in EC-8 for re-entrant corners has been satisfied.

ii. Check for slenderness of structure
   The program calculates a “slenderness ratio” for the structure based on the maximum dimensions on plan and reports whether the EC-8 criteria are satisfied.

iii. Checks for torsional radius
The program calculates the torsional radius for each floor and checks against the conditions set out in EC-8. These are summarized in the Check Regularity in Plan dialog and the Torsional Radius Check table will display details.

Opens when the Eurocode 8 | Plan Regularity page is selected.

P. Torsional Radius Check table

The program calculates the torsional radius for each floor and checks against the conditions set out in EC-8.

Opens when the Eurocode 8 | Plan Regularity page is selected.

P. Slab Re-Entrant Corners table

The program performs checks on each floor slab and reports whether the condition set out in EC-8 for re-entrant corners has been satisfied. The details of any corners detected which fail this check are displayed here. The geometry which fails is highlighted in the active view window.

Opens when the Eurocode 8 | Plan Regularity page is selected.

P. Eurocode 8 | Elevation Regularity page

Used to classify the building as a “regular in elevation” structure. Checks will be performed and the results will be displayed graphically.

When the Eurocode 8 | Elevation Regularity page is selected, the Earthquake Elevation Criteria dialog and Elevation Regularity table open.

The structure diagram is displayed in the Whole Structure view window. When a Elevation Regularity check is performed, the elevation profile is displayed on the structure diagram.

P. Earthquake Elevation Criteria dialog

Used to specify global seismic direction and the elevation criteria per Eurocode 8 against which a Elevation check will be performed.

Opens when the Eurocode 8 | Elevation Regularity page is selected.

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Choose elevation along</td>
<td>Select the global axis along which the seismic forces are acting for this check.</td>
</tr>
<tr>
<td>Choose elevation criteria</td>
<td>Select the criteria which best describes the elevation variation per Clause 4.2.3.3 and Figure 4.1 of EN 1998-1:2004 (EC8).</td>
</tr>
<tr>
<td>Perform Checks</td>
<td>Click this button once the appropriate criteria is selected for a direction. The program will evaluate the elevation geometry against code requirements. The results of this evaluation will populate the Elevation Regularity table and the profile checked will be displayed onto the structure diagram in the active view window.</td>
</tr>
</tbody>
</table>

P. Elevation Regularity table

Reports the status of a Elevation Regularity check performed based on criteria set in the Earthquake Elevation Criteria dialog.
Opens when the Eurocode 8 | Elevation Regularity page is selected.

P. Plotting from STAAD.Pro

Explained below are five methods for plotting the drawing of the STAAD model and STAAD result diagrams.

P. Plot Using the Print Current View Tool

1. (Optional) Right-click in the view window and then select the Print Preview Current View from the pop-up menu.
   
   This is used to preview the image as it will appear on paper.
   
   The Print Preview window opens to display the view in a report format.

   2. Select Print in the Print Preview toolbar.

   The standard Windows Print dialog opens, which is used to select the printer or plotter where you wish to send the image.
3. Select the desired printer and click **OK**.

**P. Plot Using the Take Picture Tool**

1. On the **Utilities** ribbon tab, select the **Take Picture** tool in the **Utilities** group.

   ![Take Picture Tool](image)

   The **Picture #** dialog opens.

   ![Picture 1](image)

   *Figure 202: The Picture # dialog with the ID and Capture automatically generated*

2. (Optional) Type a Picture **ID**

   The ID is incremented automatically.

3. (Optional) Type a **Caption**
Note: For most View window contents, the caption will be automatically completed with a description of the contents.

4. Click OK.
   This picture is saved for use in reports.

5. Open the Report Setup:
   a. Select the File ribbon tab.
      The Backstage view opens.
   b. Select the Report tab and then Setup.

   The Report Setup dialog opens.

   ![Report Setup dialog](image)

   Note: The Available items list is filtered by the drop-down selection list above it. This allows you to sort your reports.

6. On the Items tab, select Pictures from the available list of items.
7. Add one or more pictures to the Selected list by clicking >.
8. (Optional) Select the Picture Album

   Tip: You can manage pictures taken using the Take Picture tool here.
9. (Optional) Select the **Full Page**

10. Click **OK**.

11. Select **File > Print Report**.
    
The diagram will be plotted.

Alternatively, Select **File > Export Report > MS Word File** to create a report you can print from Microsoft Word.

In the **Save As** dialog, specify a name for the file and click **Save**. In the template dialog, select the **Normal** template (or use any custom Word templates you have created).

Microsoft Office Word builds the file with the picture in it. Once this task is completed, the new file is opened in Microsoft Office Word.

### P. Plot Using the Export View Option

STAAD.Pro has a facility to export the drawing to a graphic image file.

Use the program to generate the diagram or plot you want to export.

1. Select the **Export View** tool on the **Print** toolbar.
   
The **Save As** dialog opens.
2. Select the graphic format you want to use for your image file (i.e., Bitmap, JPEG, TIFF, GIF, etc.)
3. Type a file name and save the file.
P. Plot Using the Copy Picture Option

1. Highlight the window containing the diagram that you wish to plot. This can be done by making sure the title bar of that window has the color which indicates that it is in focus.
2. Select **Edit > Copy Picture**.
3. Open a graphics program capable of cropping the graphics image to your needs (e.g., Microsoft Paint, Corel PaintShop Pro, Adobe Photoshop, etc.).
4. Paste the clipboard contents into an empty file in this program. The drawing from the STAAD window is pasted in that program.

Alternatively, you can also copy graphic contents of the View window by using the `<PrtSc>` (Print Screen key).
This section describes how to manually edit STAAD.Pro input files, import and export data, and interact with STAAD.Pro using external applications.

**Note:** You can also interact with STAAD.Pro via an API. Refer to [OpenSTAAD](on page 5045) for more information.

### I. STAAD.Pro Editor

This section describes the various facilities available in the STAAD.Pro Editor. The tools and menus in the editor are described here.

#### Related Links
- [P. To view analysis results](on page 2212)

### I. Getting Started

### I. Quick Overview

- **File tab**
  - All file management tools are here

- **Contents & Bookmarks**
  - File navigation is included here

- **Input file view**
  - Type and edit input files on each tab in this area

- **Error list**
  - Messages, warnings, and errors detected in the input file display here

- **Quick access toolbar**
  - Add commonly used tools for your convenience

- **Status bar**
  - Displays length of input file and cursor position

- **Home tab**
  - Editing, navigation, and other tools are here

- **Find & Replace**
  - Search for and replace text strings

**Related Links**

- [I. File tab](on page 2262)
- [I. Home tab](on page 2263)
I. To start the editor from STAAD.Pro

When the STAAD.Pro Editor is opened from within STAAD.Pro, the model that is currently open is saved, the STAAD input file automatically parsed and a check on the data is performed. If any issues detected are then reported in an error message that identifies the number of Errors/Warning detected. The details of the issues are then reported in the Error List panel. For more details for the messages. Refer to I. To check for syntax errors (on page 2256) for details.

**Note:** Note that when the STAAD.Pro Editor is opened from within STAAD.Pro, it is not possible to open an additional STAAD input file or start a new file. This is only possible when starting the STAAD.Pro Editor externally. Refer to I. To start the editor externally (on page 2252).

1. Open a STAAD file in STAAD.Pro.
2. On the **Utilities** ribbon tab, select the **Command File Editor** tool in the **Utilities** group.

The STAAD.Pro Editor window opens with the current STAAD input file.

I. To start the editor externally

STAAD.Pro Editor can be run separately from STAAD.Pro.

**Note:** STAAD.Pro Editor is not capable of checking for errors when run externally from STAAD.Pro. Additionally, STAAD.Pro Editor cannot perform an analysis on an input file. This must always be done using STAAD.Pro.

1. In Windows Explorer, navigate to the Editor folder in the STAAD.Pro installation folder.
   This is typically *C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\Editor*.
2. Either:
   - double-click the *Bentley.Staad.Editor.exe* or *Bentley.TCL.Editor.exe* file
   - or
   - drag-and-drop a STAAD input file (file extension .std) onto the STAAD.Pro Editor icon.

**Tip:** You can create a Windows shortcut to the STAAD.Pro Editor program file and add it to your Start menu, desktop, or Windows task bar.

**Tip:** You can right-click on an .std and select **Open with** to add the STAAD.Pro Editor to the program list associated with a STAAD input file.
I. To Pin a Panel

The Contents, Bookmarks, Find and Replace, and Error List panels call be “pinned” to stay open. Alternatively, to free up screen space, they can be “unpinned” to display only when selected.

1. Either:
   - click the tab name of the panel you want to display
   - or
   - click the corresponding tool name in the Searching or View groups on the Home ribbon tab
     The panel opens.

2. Click the pin icon in the top, right-hand corner of the panel.
   The panel is now pinned in place.

   **Tip:** Clicking the pin icon again will hide the panel once you click elsewhere.

**Related Links**

- I. Bookmarks (on page 2257)

I. STAAD Input Files

These are text files which list structure geometry, material properties, loading data, analysis commands, design commands, and other related structure input data which the STAAD.Pro analysis and design engine interprets upon running an analysis.

These files are the primary files created in the STAAD.Pro graphical interface and can be edited directly using the STAAD.Pro Editor.

   **Tip:** STAAD input files use the .std file extension.

**STAAD Command Language and Syntax**

The advantage of using the STAAD.Pro Editor over other plain text editors is that the syntax of the STAAD command language is natively understood and will be appropriately marked (this functionality may be customized in the STAAD.Pro Editor Settings dialog). The following represents an overview of the language structure:

- English based - Input commands use full English words (though these may be shortened in many instances) or common engineering nomenclature.
- Case insensitive - Input data is not case sensitive.
- Line continuation - Lines containing lists may be continued to the next line by ending the line with a blank and a hyphen. Some lists have special rules or considerations.
- Comments - Lines may be marked as a comment (information which is ignored by the program) by including an asterisk (*) as the first non-blank character in any line. By default, comment lines will appear in green in the STAAD.Pro Editor window.

Refer to Section 5.1 in the Technical Reference manual (on page 2405) for additional information on the STAAD Command Language.
I. To create a new STAAD input file

1. Either:
   Select the New tool on the File ribbon tab.
   or
   Press <Ctrl+N>.
   A new STAAD input file opens with the empty template.

You can now start typing STAAD commands. To check syntax or to run the STAAD input file, you must use STAAD.Pro.

I. To open an existing STAAD input file

Tip: Multiple input files can be open simultaneously. Each file is displayed on a separate tab across the top of the text editor.

1. Either:
   Select the Open tool on the File ribbon tab.
   or
   Press <Ctrl+O>.
   The Select STAAD File dialog opens. The controls are similar to a typical Windows File open dialog.
2. Navigate to and select the STAAD input file you want to edit.
3. Click Open.

I. To save changes to a STAAD input file

1. Either:
   select the Save tool on the File ribbon tab
   or
   press <Ctrl+S>.

   Tip: Select the Save As tool on the File ribbon tab to save the file with a different file name or in a different location.

   If this is a newly create STAAD input file, you will be prompted to type a file name and select a location to save the file.
2. (Optional) Click Save if you are saving for the first time or saving as a new file.

I. Getting Help

When you are typing, press <F1> to open the Technical Reference help to the topic pertaining to the current input command.

To open the general STAAD.Pro help, click the Help tool ( ) on the right side of the ribbon bar.
I. Editing Input Files

I. Typing Commands

Code Completion

As you type commands, the STAAD.Pro Editor will offer suggestions to complete the command you are typing in the form of a drop-down list. This list is context-sensitive based on the preceding commands.

As you continue typing, the list will respond by narrowing to complete the word you are typing. You can use your mouse or the arrow keys to select a command from the list. Press <Enter> or <Tab> to accept the highlighted command keyword.

Table Sizes

Similarly, within a MEMBER PROPERTY country block, the corresponding country section tables are available for selecting sections. After typing TABLE and any additional specification options, the table sections will open as a drop-down list. You can continue typing to narrow the list or use your mouse to select one of the shape tabs for that table.

Context-Sensitive Command Help

Pressing <F1> opens the Technical Reference help to the topic corresponding to the current input command.

I. To add a comment

To change one or more lines in the input file to a comment, do the following.

Comments can be used to add notes or remarks in the input. They also can be used to turn 'off' commands so they are ignored by the analysis and design engine.

1. Either:
   - place your cursor anywhere on the line you want to comment
   - or
   - select text across multiple lines

2. Either:
   - select the Comment tool in the Editing group on the Home ribbon tab
   - or
   - right-click and select Comment Selection

   If... Then...
   No content was selected in Step 1 the current line is made a comment
   Multiple lines were selected in Step 1 all the lines in the selection are made comments
**Tip:** You can remove comments using the **Uncomment** tool in the **Editing** group on the **Home** ribbon tab or by selecting **Uncomment Selection** in the right-click menu.

I. To check for syntax errors

**Note:** In order to check for errors, STAAD.Pro must be running. If you have launched the STAAD.Pro Editor externally, this feature is disabled.

1. Select the **Check Syntax** tool in the **Tools** group on the **Home** ribbon tab. A message opens to indicate if no errors are detected.

You can review individual errors or warnings in the **Error List** panel.

I. Navigation

Multiple Files Open

You can open multiple STAAD input file simultaneously. Each file is displayed on a separate tab across the top of the text editor panel by the filename. The currently selected tab is active and all operations are performed on this file only.

Blocks

Blocks can be nested. For example, `DEFINE MATERIAL START` is a block, and each `ISOTROPIC` material within that block is a sub-block. The current block is indicated by a vertical bar to the right of the commands.

Clicking the minus sign will collapse the block down to the fist command. Once a block is collapsed, a plus sign is displayed and the first command is displayed in a box.

**Tip:** Select the **Collapse** tool in the **Outlining** group on the **Home** ribbon tab to collapse all blocks in the current input file.

Contents Panel

The top-level blocks are listed, in order, in the Contents pane.

Double-clicking an entry in the contents moves the cursor to that point in the text editor.

I. To go to a line

To move the cursor to a particular line in the input file, do the following:

1. Select the **Go to** tool in the **Searching** group on the **Home** ribbon tab. The **Go To Line** dialog opens.
2. Type the **Line Number**.
3. Click **OK**. The cursor is placed at the beginning of the indicated line.

**Note:** A message indicates if the specified line number is higher than the number of lines in the file.
I. Bookmarks

You can add bookmarks at any point in an input file to quickly return to that point.

**To add a bookmark**

1. Either:
   - select the Add tool in the Bookmarks group on the Home ribbon tab, or
   - right-click and select Toggle Bookmark from the pop-up menu.

To navigate to the next or previous bookmark in the file, select the Next or Previous tool in the Bookmarks group on the Home ribbon tab.

To jump to a specify bookmark, double-click the name in the Bookmark panel.

To remove all the bookmarks in the file, select the Clear tool in the Bookmarks group on the Home ribbon tab.

**Related Links**
- *I. To Pin a Panel* (on page 2253)

---

I. Find and Replace

**I. To find something**

1. Either:
   - select the Find tool in the Searching group on the Home ribbon tab
   - or
   - press `<Ctrl+F>`

The Quick Find tab on the Find and Replace pane opens.

2. Type a string to match in the Find what field.

3. (Optional) Select the Look in and Find options to specify the extents of the search and how the string should be matched, respectively.

4. Click Find Next.

The next matching string is selected in the input file.

**Related Links**
- *I. Search Methods* (on page 2258)

---

I. To replace something

1. Either:
   - select the Replace tool in the Searching group on the Home ribbon tab
   - or
   - press `<Ctrl+H>`

The Quick Find tab on the Find and Replace pane opens.

2. Type a string to match in the Find what field.
3. (Optional) Select the **Look in** and **Find options** to specify the extents of the search and how the string should be matched, respectively.

4. Type a replacement string in the **Replace with** field.

5. Either:
   - click **Find Next** to highlight the next matching string
   - or
   - click **Replace** to replace the currently highlighted match
   - or
   - click **Replace All** to automatically replace all the matching strings in the file

**Related Links**
- *I. Search Methods* (on page 2258)

---

### I. To find all occurrences of a string

1. Press `<Ctrl+I>`.
2. Start typing a string.
   - As you type, all occurrences of the input string are highlighted in the file. The cursor jumps to the next occurrence of the string.

---

### I. Search Methods

STAAD.Pro Editor can use a variety of methods for matching strings when searching an input file.

In either the **Quick Find** or **Quick Replace** tabs on the **Find and Replace** panel, expand the **Find options** section to display the available matching options.

- **Look in** Select either the **document** (entire file) or **selection** to limit the search extents.
- **Match case** Select use case-sensitive matching in the search string.
- **Match whole word** Select to match entire words (separated by white space) only. (e.g., searching for “JOI” will find “JOI” but not “JOINTS”).
- **Search type**
  - **Normal** Match the input string literally, using the options selected above.
  - **Regular Expressions** Use regular expressions (also referred to as regex or regexp) in the search string. For additional
  - **Wildcard** Use standard Windows wildcards to substitute a single character for one or more other characters in the search string.
    - “?” represents a single character, and
    - “*” represents any combination of characters

- **Acronym**
- **Shorthand**

**Related Links**
- *I. To find something* (on page 2257)
I. Snippets

Code snippets are short sections which you commonly need to insert into different input files. Saving this code as a separate snippet allows you to easily access these portions of predefined code.

I. To insert a code snippet

1. Place your cursor at the point you want to insert a code snippet.
2. Right-click and select **Insert Snippet** from the pop-up menu.

   The inline snippet tool opens.

3. Either:

   Use the arrow keys or mouse to select the name of the snippet you want to insert.

   **Tip:** If you have organized snippet files into multiple folders, select the folder first.

   or

   Start typing to select the snippet by the Shortcut string in the snippet file.

4. Double-click with the mouse, press **<Enter>**, or press **<Tab>** to select and place the snippet.

The text is inserted.
I. To create a code snippet

To create a custom code snippet using a sample as a template, do the following:

Some familiarity with editing XML files (e.g., HTML) is helpful to create snippet files, but not necessary. Further, no special tools or software are required, but a plain text editor which can validate XML is recommended.

1. In Windows Explorer, navigate to the .../STAAD/Editor/Snippets/Std/ folder in your STAAD.Pro installation folder.
2. Make a copy of one of the existing snippet files (file extension .snippet).
   You can place snippets in the same folder or create a new folder within .../STAAD/Editor/Snippets/ to organize snippet files as needed.
3. Rename this copy to something meaningful.
4. Open the renamed file in a plain text editor.
5. Change the header information:
   a. Type a new title within the <Title> element.
   b. (Optional) Type a your name or your organization’s name within the <Author> element.
   c. Type a tool tip description within the <Description> element.
   d. Type a shortcut string within the <Shortcut> element.

   **Note:** Do not edit any of the XML tags or attributes. Only change the plain text inside elements.

6. (Optional) If you want to add replacement strings in the snippet, create a <Literal> element inside the <Declarations> section for each.
7. Type or paste the actual snippet STAAD input code into the CDATA section, between the inner-most square brackets.
   Example:
   
   ```
   <![CDATA[snippet contents here]]>
   ```
   The contents here will be copied directly when the snippet is used (except for variables), including line breaks and commented lines.

   **Tip:** There is no syntax checking in the snippet file itself, so it is best practice to copy and paste the STAAD input snippet code from a STAAD input file you have checked for errors.

8. Save and close the snippet file.

I. Creating a New Code Snippet

Code snippet files (file extension .snippet) are XML files which contain one or more code snippets to insert along with metadata about the snippet, such as tool tips, titles, etc. They can be created or edited in any plain-text editor.

   **Tip:** It is easiest to copy an existing snippet file, rename it, and edit as needed.

You can save it in a separate folder to organize your snippets. This will add a folder section to your Insert Snippet menu. Otherwise, you can simply save them in the /STAAD/Editor/Snippets/Std/ folder.

The application comes with a few sample snippet files in the installation. However, to add your own Snippets, add the .snippet file to the folder

\Users\(user name)\AppData\Local\Bentley\Engineering\STAAD.Pro CONNECT Edition\Default \Editor\Snippets.
When you select Insert Snippet from the right-click pop-up menu, you will then be provided folders for the samples from Bentley as well as those you have created in the your user folder.

**Code Snippet Syntax**

When editing a snippet file, you should note that there are two primary components to the XML: the `<Header>` and the `<Snippet>`. The Header contains the metadata about the code snippet, including the `<Title>` text and `<Description>`, which is used as a tool tip in the STAAD.Pro Editor.

The actual code inserted when the snippet is used by the STAAD.Pro Editor is contained in a CDATA section within the `<Code>` section.

```xml
<Code Language="STD">
  <![CDATA[UNIT FEET KIP]]>
</Code>
```

**Tip:** If you want line returns before or after the inserted code, you can place those at the beginning or end of the CDATA text. Whitespace (i.e., spaces, tabs, or line returns) outside of the CDATA section have no effect on the inserted snippet.

```xml
<Code Language="STD">
  <![CDATA[
     UNIT FEET KIP
  ]]]>
</Code>
```

**Snippet Replacements**

You can create replacement strings in a snippet which will be highlighted for the snippet user. These are added in the `<Declarations>` section within the `<Snippet>` section.

Each replacement definition has the following structure:

```xml
<Literal>
  <ID>name</ID>
  <ToolTip>Tool tip text.</ToolTip>
  <Default>Default text</Default>
</Literal>
```

Where:

- `name` is the replacement name to use within the snippet code.
- `Tool tip text` is the string to show as a tool tip in the STAAD.Pro Editor
- `Default text` is the text that will be inserted (and highlighted) in the STAAD.Pro Editor when the snippet is used

To use the replacement within the snippet code, just place dollar signs ("$") before and after the `name` value.
Snippet Editing Tools

You can use an IDE such as Microsoft Visual Studio, but any plain text editor is capable of editing code snippet files. An editor that is capable of checking XML syntax is recommended, though.

Tip: Notepad++ is a free text editor which, when the XML Tools plugin is installed, is capable of checking XML syntax.

Related Links
- MSDN Walkthrough: Creating a Code Snippet

I. Ribbons

I. File tab

Table 212: Controls on the File ribbon tab

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>What it Does</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="New" /></td>
<td>Creates a new STAAD input file.</td>
<td>&lt;Ctrl+N&gt;</td>
</tr>
<tr>
<td><img src="image" alt="Open" /></td>
<td>Opens an existing STAAD input file.</td>
<td>&lt;Ctrl+O&gt;</td>
</tr>
<tr>
<td><img src="image" alt="Close" /></td>
<td>Closes the current STAAD input file (current tab only).</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Save" /></td>
<td>Saves any changes made to the current STAAD input file since the last save. These changes are highlighted with yellow bars in the text view. After saving, these changes are marked green.</td>
<td>&lt;Ctrl+S&gt;</td>
</tr>
</tbody>
</table>
## Tool Name

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>What it Does</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Save As</td>
<td>Opens a <strong>Save As</strong> dialog, which is used to save the current STAAD input file with a different name or in a different location.</td>
<td></td>
</tr>
<tr>
<td>Print Preview</td>
<td>Opens a <strong>Print Preview</strong> window, which is used to review the print of the file.</td>
<td></td>
</tr>
<tr>
<td>Print</td>
<td>Opens a <strong>Print</strong> dialog, which is used to select a printer device and print out a copy of the file.</td>
<td>&lt;Ctrl+P&gt;</td>
</tr>
<tr>
<td>Send in email</td>
<td>Creates a draft e-mail in your default e-mail client with the current STAAD input file attached. The file path to the input file is inserted in the body of the message.</td>
<td></td>
</tr>
<tr>
<td>About</td>
<td>Opens the <strong>About</strong> dialog, which contains version and copyright information for the program.</td>
<td></td>
</tr>
<tr>
<td>Exit Editor</td>
<td>Closes the STAAD.Pro Editor program.</td>
<td>&lt;Alt+F4&gt;</td>
</tr>
</tbody>
</table>

### Related Links
- [I. Quick Overview](on page 2251)

### I. Home tab

**Tip:** Double-click the **Home** ribbon tab or select the ^ button to collapse the ribbon. Clicking the tab again displays the toolbar.
Table 213: Clipboard group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Paste</td>
<td>Pastes the clipboard contents (text only) at the cursor position.</td>
<td>&lt;Ctrl+V&gt;</td>
</tr>
<tr>
<td>Cut</td>
<td>Copies the selected contents to the clipboard and deletes the original.</td>
<td>&lt;Ctrl+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copies the selected contents to the clipboard.</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
</tbody>
</table>

Table 214: Editing group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo</td>
<td>Undoes the last action in the editor.</td>
<td>&lt;Ctrl+Z&gt;</td>
</tr>
<tr>
<td>Redo</td>
<td>Reverses the last undo action.</td>
<td>&lt;Ctrl+Y&gt;</td>
</tr>
<tr>
<td>Comment</td>
<td>Changes the current line (or lines, if a selection spans multiple lines) to a comment (i.e., adds an asterisk (*) at the beginning of the line so it is ignored by STAAD.Pro).</td>
<td></td>
</tr>
</tbody>
</table>
## Data Files and Interoperability

### I. STAAD.Pro Editor

### Table 215: Searching group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uncomment</td>
<td>Changes a commented line or lines back to commands to be interpreted by STAAD.Pro.</td>
<td></td>
</tr>
</tbody>
</table>

### Table 216: Bookmarks group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>Adds a bookmark to the line where the cursor is located.</td>
<td>&lt;Ctrl+F2&gt;</td>
</tr>
<tr>
<td>Clear</td>
<td>Clears all bookmarks from the current input file.</td>
<td></td>
</tr>
</tbody>
</table>
# Data Files and Interoperability

## I. STAAD.Pro Editor

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Previous</td>
<td>Jumps the cursor to the previous bookmark, if present, in the current input file.</td>
<td>&lt;Shift+F2&gt;</td>
</tr>
<tr>
<td>Next</td>
<td>Jumps the cursor to the next bookmark, if present, in the current input file.</td>
<td>&lt;F2&gt;</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Collapse</td>
<td>Collapses all top-level blocks in the text editor for the current input file.</td>
</tr>
<tr>
<td>Expand</td>
<td>Expands all blocks in the text editor for the current input file.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td>Opens then <strong>Contents</strong> panel.</td>
</tr>
<tr>
<td>Error List</td>
<td>Opens then <strong>Error List</strong> panel.</td>
</tr>
</tbody>
</table>
### Table 219: Tools group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bookmarks</td>
<td>Opens then <strong>Bookmarks</strong> panel.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Check Syntax</td>
<td>Checks for errors in the current input file. Any errors, warnings, or messages are displayed in the <strong>Error List</strong> panel.</td>
</tr>
<tr>
<td>Converter</td>
<td>Opens then <strong>STAAD.Pro Unit Converter</strong> program, which can be used to convert a variety of common units.</td>
</tr>
<tr>
<td>Calculator</td>
<td>Opens then <strong>Calculator</strong> program, which can be used for basic calculations.</td>
</tr>
</tbody>
</table>

### Table 220: Misc controls on the Home ribbon tab

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Settings</td>
<td>Used to change settings in the STAAD.Pro Editor program.</td>
</tr>
<tr>
<td>Always On Top</td>
<td>Used to keep the STAAD.Pro Editor in the foreground, in front of all other Windows (regardless of focus). This feature is active when the icon is highlighted. Select again to deactivate.</td>
</tr>
</tbody>
</table>
I. Settings dialog

This dialog is used to control display settings for the STAAD.Pro Editor.

General tab

Display Line Number  Set this option to include line numbers to the left of the command lines.
Highlight Current Line Set this option to highlight the entire line where the cursor is located.

Fonts and Colors tab

Font Family  Select an installed Windows font from the drop-down list to use for the text editor.
Font Size  Select a font size from the drop-down list to use for the text editor.
Display Items  Select one of the text window display items from this list. The color can be selected from the Item Color drop-down pallet.
Item Color  Click the down arrow to select a color for the selected Display Item.

Print tab

Print Document Information  Set this option to include the file path, file name, and timestamp of the file in the footer of each page of the printout.
Print Job Information  Set this option to include the JOB NAME, JOB CLIENT, and ENGINEER NAME job information values from the START JOB INFORMATION command block.
Print Page Number  Set this option to include page numbers in the footer of each page of the printout.
Print Company Logo  Set this checkbox to include the file path in the Company Logo File field. Click Browse to locate an image file.

I. Keyboard Shortcuts

<table>
<thead>
<tr>
<th>Action</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exit the programs.</td>
<td>&lt;Alt+F4&gt;</td>
</tr>
<tr>
<td>Opens the help topic for the current input command.</td>
<td>&lt;F1&gt;</td>
</tr>
<tr>
<td>Select all the contents in the current input file.</td>
<td>&lt;Ctrl+A&gt;</td>
</tr>
<tr>
<td>Copy the current selection to the clipboard.</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
<tr>
<td>Open the Quick Find tool to search the document.</td>
<td>&lt;Ctrl+F&gt;</td>
</tr>
<tr>
<td>Open the Quick Replace tool to find and replace strings in the document.</td>
<td>&lt;Ctrl+H&gt;</td>
</tr>
<tr>
<td>Highlight all occurrences of a typed string.</td>
<td>&lt;Ctrl+I&gt;</td>
</tr>
<tr>
<td>Delete the current line.</td>
<td>&lt;Ctrl+L&gt;</td>
</tr>
<tr>
<td>Start a new input file.</td>
<td>&lt;Ctrl+N&gt;</td>
</tr>
</tbody>
</table>
Data Files and Interoperability

I. Integrated Structural Modeling

I. ISM Sync Tools Overview

STAAD.Pro can send structural data to and from an ISM repository through a set of ISM Syncing tools. These tools allow both creation and updating of STAAD.Pro models as well as ISM repositories. Further, these flexible tools allow you to begin models and move data as your workflow dictates.

<table>
<thead>
<tr>
<th>If you need to ...</th>
<th>... use this tool</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create a new ISM repository from an existing STAAD.Pro model</td>
<td>Create ISM Repository</td>
<td>transfers the current model opened in STAAD.Pro and generates a new ISM repository. This is the most common way in which an ISM repository is initially created.</td>
</tr>
<tr>
<td>Create a new STAAD.Pro model from an existing ISM repository</td>
<td>New From ISM Repository</td>
<td>creates a new STAAD.Pro model from an existing ISM repository. This is used to transfer model data from other tools used in the workflow.</td>
</tr>
</tbody>
</table>
If you need to ... | ... use this tool | Description
---|---|---
Update an existing repository to reflect changes made in a STAAD.Pro model | Update ISM Repository | will coordinate changes made to the model in STAAD.Pro and coordinate some or all of those changes with an existing ISM repository.
Update an existing STAAD.Pro model to reflect changes in an ISM repository | Update From ISM Repository | allows the user to update a STAAD.Pro model with some or all of the changes which have been made to the ISM repository.

Not all tools are available at all times. For example,
- If ISM is not installed these items will be grayed out and not available.
- If ISM is installed but no STAAD.Pro model is loaded, only New From Repository will be available.

Structural Synchronizer

The program provided by ISM for accepting or rejecting model data changes is called "Structural Synchronizer". Structural Synchronizer provides the user with a powerful set of tools for moving data between the applications used in a daily workflow. Even relatively small structural models have enormous amounts of data and ISM allows this data to be re-used with ease. Care must be taken that only desired data is transferred between applications.

When accepting changes made by client applications to an ISM Repository, some attention must be paid to what changes are actually being made. A small change in a client application can have unintended repercussions, if accepted. A repository is intended to represent the data that is common to all the client applications. Some client application models will use only a subset of the repository data but changes made to them can affect the entire repository if these changes are accepted when a repository update action is performed.

**Note:** This brief overview is not intended as a sufficient description on using the full capabilities of Structural Synchronizer. Please refer to the Structural Synchronizer product documentation for more detailed information.

I. What is ISM?

Bentley’s Integrated Structural Model (ISM) is a technology for sharing structural engineering project information among structural modeling, analysis, design, drafting, and detailing applications. ISM is similar to Building Information Modeling (BIM), but focuses on the information that is important in the design, construction, and modification of the load bearing components of buildings, bridges, and other structures.

I. Purpose of ISM

There are two related purposes for ISM:

1. The transfer of structural information between applications.
2. The coordination of structural information between applications.

To provide for the first purpose (transferring information), ISM provides a means of defining, storing, reading and querying ISM models.

To provide for the second purpose (coordination of information), ISM additionally provides capabilities to detect differences between ISM models and to selectively (based on user selection) update either an ISM repository or an application’s data to provide a user-controlled level of consistency between the two data sets.

I. ISM and Application Data

ISM is not intended to store all of the information that all of its client applications contain. Rather, it is intended to store and communicate a consensus view of data that is common to two or more of its client applications, such as STAAD.Pro.

STAAD.Pro continues to hold and maintain its own private copy of project data. Some of the application data will duplicate that of the associated ISM repository. The application data may even conflict with that in the ISM repository. STAAD.Pro (or the user) may decide that a conflict gives the best data for the different uses in STAAD.Pro versus ISM.

I. Backups

STAAD.Pro has an auto-saving and backup managing feature to prevent accidental data loss.

I. To enable auto save

To turn on the auto save feature and set the frequency of

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Select the **Tools** tab and then click **Open Backup Manager**.

   ![Backup Manager dialog](image)

   The **Backup Manager** dialog opens.
3. Check the option **Enable Auto Save**.
4. (Optional) Type the how often (in minutes) you want to be prompted to save changes in the **Frequency of Auto Save** field.
5. Click **OK**.

I. Archives

Archive files can be used to collect all of the files within a STAAD.Pro into a single file, which is useful for records or sending project files to others.

STAAD.Pro projects consist of multiple files, such as the input file (file extension .std), analysis results (file extension .anl), etc.
An archive file uses the extension .stz.

Related Links
- I. To share a STAAD.Pro project in ProjectWise (on page 2285)

I. To create an archive

To create an archive file for a STAAD project, use the following procedure.
You cannot create an archive when a STAAD input file is open in the program.

1. Select the Archive tab on the Start page.
2. Select Create.

3. Select the File Name of the STAAD input file you want to archive.
   Click Browse to navigate to the file using a Windows open dialog.
4. Type the File Name of the archive file.
5. (Optional) Select a Location where you want the archive file created.
   The default location is the same folder containing the STAAD input file.
6. (Optional) Select the files associated with the input file you want included in the archive.
   All files are selected by default.
7. Click Create.

You can repeat this process and overwrite an existing archive file to update with any changes made to the
STAAD input file, analysis results, etc.

Related Links
- GS. Start Page (on page 50)

I. To open an archive file

To open an archive file in STAAD.Pro, use the following procedure.

1. Select the Open tab on the Start page.
2. Select the Archive tab as the source type.

Tip: You can open recent Archive files with one click. These are listed on the Archive tab along the top of the
Open tab

The Windows Open dialog opens.

3. Select the File Name of the archive file (file extension .stz) you want to open.
Tip: Click the […] button to navigate to the folder where the archive is saved.

4. Select the Location where you want to store the extracted project files.
5. Click Open.

You can work with the STAAD input file and associated files within the archive as you normally would. Select File > Close to close the archive when you are finished.

Related Links
• GS. Start Page (on page 50)

I. To extract an archive

To extract the files within an archive, use the following procedure.

You cannot extract an archive when a STAAD input file is open in the program.

1. Select the Archive tab on the Start page.
2. Select Create.

3. Select the File Name of the archive file you want to extract.
   Click Browse to navigate to the file using a Windows open dialog.
4. (Optional) Select a Location to where you want the files within the archive extracted.
   The default location is the same folder containing the archive file.
5. (Optional) Select the files in the archive you want to extract.
   All files are selected by default.
6. Click Extract.

Related Links
• GS. Start Page (on page 50)

Bentley CONNECT Features

ProjectWise Project Association

STAAD.Pro CONNECT Edition allows you associate a file with a ProjectWise Project.

A ProjectWise Project is a single definition of a project for your entire organization and represents a one-to-one relationship with the contracted work being done by your organization.

Note: In order to utilize this feature in STAAD.Pro, you must:

1. Have the Bentley CONNECTION Client running. The CONNECTION Client is typically installed with STAAD.Pro.
2. Register with Bentley Cloud Services.
3. Sign in using your credentials with the CONNECTION Client.

For additional details on the benefits of using ProjectWise Projects, please visit https://www.bentley.com/connect.

Related Links
- GS. To create a new STAAD.Pro model (on page 33)
- GS. To open a STAAD.Pro model (on page 34)

To Associate a ProjectWise Project with Your File

When you create a new file or open an existing file which is not associated with a project, use the following procedure to associate your file with a ProjectWise Project.

**Note:** You must be signed in using the CONNECTION Client to associate a ProjectWise Project with your file.

**Tip:** If you want to change the ProjectWise Project associated with your file, use the same following procedure.

1. On the **File** ribbon tab, select, **Cloud Services > Associate Project**

   The **Assign Project** dialog opens.

   ![Assign Project dialog](image)

   **Project List**
   - Number: 10214-23-237
   - Name: C&C Machines
   - Location: Nashville
   - Industry: Buildings and Facilities
   - Asset Type: Healthcare Facilities

   - Number: 00002-DS-43458
   - Name: I-64 Phase 1
   - Location: St Louis, Mo
   - Industry: Roads and Highways
   - Asset Type: Bridges

   - Number: NC-1710
   - Name: Bridge design project
   - Location: New York
   - Industry: Roads and Highways
   - Asset Type: Bridges

   - Number: F9H615698
   - Name: F9H615698
   - Location: F9H615698
   - Industry: Buildings and Facilities
   - Asset Type: Airports

   - Number: DLG1
   - Name: DLG project 1
   - Location: Exton
   - Industry: Mapping and Surveying
   - Asset Type: Other

2. (Optional) If you want to register a new project, do the following:
   a. Click **Register Project**.

      The **Register a Project** page opens in your browser.

      **Note:** Only users with Admin/Co-admin roles can register a project.

   b. Type or select the required items (marked with an asterisk, "*")
   c. Click **Save**.
A list of registered projects within your organization opens. The newly created project is highlighted in green.

See Register a ProjectWise Project (on page 2277).

**Tip:** Alternately, you can visit connect.bentley.com and select +New on the Recent Projects tile on your personal dashboard.

3. Select the desired project from the list.

**Tip:** Use the View controls and Search tool to locate your project. If your project is not in the list, you can add a project following the steps given in Register a ProjectWise Project (on page 2277).

4. Click **Assign**.

**Related Links**
- **Assign Project dialog** (on page 2275)

### To Disassociate a ProjectWise Project from a File

If you need to disassociate a file from a ProjectWise Project, select Cloud Services > Disassociate Project on the backstage view.

The project association is removed from the file.

**Tip:** If you want to change the ProjectWise Project association to another ProjectWise Project, this procedure is not necessary.

**Related Links**
- **Assign Project dialog** (on page 2275)

**Assign Project** dialog

Used to select a project to associate with your current file or model.
Register Project Opens the Register a Project page in your browser from where you can register a project.  

**Note:** Only users with Admin/Co-admin roles can register a project.

Refresh Refreshes the list of available ProjectWise Projects.

View Allows you to choose the list of projects that you want to see in the list box. Following are the options:
- Favorites - Displays the projects that are marked as favorites.
- Recent - Displays the recently used projects.
- All - Displays all the projects.

Search Searches through the list of available projects.

List box Displays the following columns:
- Favorite - Allows you to favorite a project. Select the star icon in this column for the project that you want to mark as favorite.
- Number - Displays the number of the project.
- Name - Displays the name of the project.
- Location - Displays the geographic location of the project.
- Industry - Displays the industry of the project.
- Asset Type - Displays the asset type of the project.

Associate Associates the selected project with the WorkSet.

Related Links
- To Disassociate a ProjectWise Project from a File (on page 2275)
Register a ProjectWise Project

Organizations can enable CONNECTED Users to register and collaborate on ProjectWise Projects. These projects contain project information such as Project Name, Asset Industry, Asset Type, Location etc. While creating a file in a CONNECT Edition product, you can associate it to a ProjectWise Project where the project information is included in the data files as properties.

Note: Project files, such as DGN files and library files are not stored on the cloud. They can be stored locally, on a network, or in ProjectWise.

What is the ProjectWise Project Registration Utility?

The Project Registration utility is an administrative interface for registering an Organization’s projects with Bentley. Registered projects are referred to as ProjectWise Projects. ProjectWise Projects provide information regarding the project themselves, as well as serving as a focal point for tying together other sources of project information.

For example, user and product usage for reporting and access to services available for each project.

Who can register a ProjectWise Project?

To register a ProjectWise Project, a user must have Administrator or Co-administrator privileges associated with their Bentley account. These privileges are required because registered ProjectWise Projects are Organization-wide resources that represent real-world projects and are used in many different locations for information organization and reporting. Therefore, access is limited to those members of an Organization with sufficient privileges to ensure that only recognized and permitted ProjectWise Projects be registered on behalf of an Organization.

Note: Users within the organization who were not designated as an Administrator or Co-Administrator who are requesting rights should contact their organizations Administrator. Bentley does not fulfill these requests.

To Register a ProjectWise Project

The Project Registration utility is used to provide information about a project as well as manage previously registered projects.

Note: Only users with Admin/Co-admin roles can register a project.

1. You can register a project from either the application or from the personal portal:
   - **To use...**
   - **do this...**
     - **the personal portal** select Bentley Cloud Services > CONNECTION Center and then click the + (plus) at the top of the ProjectWise Projects panel.
     - **STAAD.Pro** select Bentley Cloud Services > Associate Project and then click + Register Project in the Assign Project dialog.

The CONNECTION Center page opens in your browser.
2. Fill out the form as needed. Required fields are marked with an asterisk ("*").

<table>
<thead>
<tr>
<th><strong>Project Number</strong> *</th>
<th>The unique project code or ID number that is officially used in your organization for internal tracking purposes. For example, DMO-063 VP 778.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Project Name</strong> *</td>
<td>The common name for the project within your organization. For example, I-565 Interchange at County Line Road.</td>
</tr>
<tr>
<td><strong>Asset</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Industry</strong> *</td>
<td>The asset industry this project belongs to. An asset industry is a group of like organizations with a common business function centered on a like set of infrastructure assets. For example, Electric Utility.</td>
</tr>
<tr>
<td><strong>Type</strong> *</td>
<td>The type of asset this project will focus on. An asset type is a set of related assets. For example, the Asset Class Electric Network is comprised of the following assets: Distribution Network, Substation, and Transmission Network.</td>
</tr>
<tr>
<td><strong>Use Location</strong></td>
<td>Displays a Location field, where you can enter the name of the project location. For example, city/state/country.</td>
</tr>
<tr>
<td><strong>Use Latitude/Longitude</strong></td>
<td>Displays the Latitude and Longitude fields, where you can enter the specific coordinates of where the project is located.</td>
</tr>
<tr>
<td><strong>Engineering Location</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Time Zone</strong></td>
<td>The time zone of the project location.</td>
</tr>
<tr>
<td><strong>Billing Country</strong> *</td>
<td></td>
</tr>
<tr>
<td><strong>Status</strong></td>
<td>The state of the project. Active means the project is open for participation. Inactive means the project is closed for participation.</td>
</tr>
<tr>
<td><strong>Allow External Team Members</strong></td>
<td>Allows the invitation process to include team members from external organizations.</td>
</tr>
</tbody>
</table>

3. Click **Save**.
   A list of registered projects within your organization opens. The newly created project is highlighted in green.

**Related Links**
- **Assign Project dialog** (on page 2275)

**Bentley CONNECT Advisor**

The Bentley CONNECT Advisor is a unified interface that enables you to view a variety of Bentley content at one location, thereby eliminating the need to browse through various sources separately. As an end user, you get the ability to browse, search, view, and interact without having to leave the product (STAAD.Pro) that you are working on.

The Bentley CONNECT Advisor scans through different sources such as Bentley Communities, Bentley LEARNserver, and Bentley YouTube channels to display relevant information with links to the web pages. For example, if you want to search for information on the Place SmartLine tool, you can enter the tool name in
Bentley CONNECT Advisor dialog’s **Search** field. You will get a list of relevant results from different locations such as forum posts, blogs and wiki posts on Bentley Communities, Bentley LEARNserver courses that discuss about the tool, and so on. You can also look out for other information such as news and announcements, upcoming events, and QuickStarts.

The Bentley CONNECT Advisor performs the following functions:

- Gathers information from the following sources:
  - Forum posts, wikis and blogs from Bentley Communities
  - Videos, Hands-on and Assessments from Bentley LEARNserver
  - Videos from Bentley YouTube channels
  - Bentley News and Announcements
  - Upcoming Bentley Events
  - Product Help
- Provides a unified interface that displays the above items gathered from their respective sites and locations
- Searches information within all the available sources (Bentley Communities, Bentley LEARNserver, Bentley YouTube channels, Bentley News and Announcements, Bentley Events, and Product Help)
- Filters information based on:
  - Product
  - Generation
  - Release Label
  - Language
  - Content type
  - Tags, region, and so on

**Note:** To be able to access the Bentley CONNECT Advisor, you need to sign into the Bentley Cloud Services using the CONNECTION Client.

**Related Links**
- *Bentley CONNECT Advisor Dialog* (on page 2279)

**Bentley CONNECT Advisor Dialog**

Used to search for content from different sources such as Bentley Communities, Bentley LEARNserver, and Bentley YouTube channels, Product Help and display relevant information with links to the web pages. You can also look out for other information such as news and announcements, upcoming events, and QuickStarts.

You can access this dialog from the following:

- **Ribbon:** Click the **CONNECT Advisor** icon on right side of the ribbon
Data Files and Interoperability

Bentley CONNECT Features

Bentley CONNECT Advisor

Search

General
Recommendations
Contextual
Expert Advisor

Suggested Trainings (19)

Before You Begin: Configuring the Dataset for a MicroStation...

This course contains instructions for installing and configuring a dataset for use with training courses related to the MicroStation CONNECT Edition.

May 24, 2018
Fundamental

Stop Creating Inefficient Models - It’s Time to Learn the Power...

The old saying, "It's never done until it's done" comes to mind when working with 3D designs. Models need to be altered and edited throughout the design process as quickly and...

Feb 15, 2018
Intermediate

3D Constraints in MicroStation CONNECT Edition

This course aims to serve both users and creators of 3D constrained models. As a user, learn to play animations based on 3D constraints and manipulate 3D constraints intuitively...

Nov 29, 2017
Advanced

Advanced Customization of the MicroStation CONNECT Edition User...

MicroStation provides for extensive customization of the user interface including custom tool boxes and tools, context menus, the ribbon, named expressions and more. Customizations...

Dec 15, 2015
Intermediate

19 items
Data Files and Interoperability
Bentley CONNECT Features

Search

Type the "search phrase" that you want to search from Bentley content. Pressing <Enter> or clicking the Search icon next to the field performs the search and displays the result from different sources. It also auto suggests strings that might have been previously searched by other users. It also supports boolean searches using AND, OR and NOT. An example of this can be as follows:

AccuSnap AND AccuDraw - Results containing both terms, MicroStation and AccuDraw, will be displayed. For terms containing more than one word, enter the words in quotes. For example, "Named Boundaries" or "Modify Solid"

Sort

Allows you to sort the results based on Author, Date, and Title. The default is Date. Sorting can be done in ascending or descending order.

Content Type

Allows you to select the source of content from which you want to get the results. Options are All, LEARN, Communities, and YouTube. By default, the content is searched from all locations. The selection in this setting is reset with change in View Criteria.

Filter

Allows you to filter the content based on Region and Tags. The selection in this setting is reset with change in View Criteria.

Knowledge Maps and Learning Paths

- Knowledge Paths - Allows you to see your progress and expertise for the product in use. It shows the status of the training that you have started/completed. It shows how you are doing in a graphical format.
- Learning Paths - Allows you to view your assigned learning paths (Self Assigned and Organizational assigned learning paths) in a tree format. It also shows the status of the trainings that you have started/completed in a particular learning path. If there are no learning paths assigned to the user, this functionality is not available.

Refresh

Allows you to reload the search results.

User Settings

Opens the User Settings dialog where you can customize CONNECT Advisor as per your preferences. You can set your desired View, Sort and Filter settings. You can do the following:
- Use last session settings - use the changes in settings saved from the last session
- Use my saved settings - save your preferred view, sort, content type and filter settings by selecting **Save current settings** and clicking **Ok**.
- Themes - select the Classic theme or the Modern theme for display of search results.
- Auto-Refresh - turn this setting on to automatically refresh the application. You can set a duration after which the application will refresh. It is Off by default.
- Keep window always on top - turn this on to keep the CONNECT Advisor dialog always on top of other application windows. It is on by default.
- Launch CONNECT Advisor on startup - This setting is inactive in STAAD.Pro CONNECT Edition Update 10.
- Restore default settings - revert to default settings.

**My Favorites**

Allows you to bookmark the content (Learn, YouTube video, Community posts, Events, News and Announcements, Product Help) of your choice as a favorite item. While viewing any content, you can right click on the desired search result and add it to your favorites list.

**About**

Opens the About Bentley CONNECT Advisor dialog where you can view the program information like version and other details about the Bentley CONNECT Advisor.

**Help**

Opens the help for the Bentley CONNECT Advisor.

**View Criteria**

Allows you to filter the results based on any one of the selected criteria. There are the following three tabs:

- General
  - Most Recent
  - Search Results
  - QuickStarts
  - Upcoming Events
  - News and Announcements

- Personal Learn - contains Learn videos and personalized suggestions when you are logged in:
  - Ongoing Training - Any Learn courses/training that you take via Learn Portal or CONNECT Advisor, will be displayed here. You can review your in-progress trainings in this view.
  - My Suggestions - Based on your in-progress trainings, CONNECT Advisor will suggest some Learn courses for you.
Data Files and Interoperability
I. Using ProjectWise in STAAD.Pro

- Contextual - content that is applicable to your current action in STAAD.Pro (for example command in progress) is displayed here.

**Result Display**
The results are listed according to the search and filter criteria. Depending on the content type, clicking a result performs the following actions:
- Bentley Communities posts - Opens the Bentley Communities page in your default browser.
- Bentley LEARNserver or YouTube videos - The selected video is played in the in-product video player.

**Message bar**
Displays information and error messages that occur in the Bentley CONNECT Advisor. It also shows the current number of results displayed.

**Related Links**
- **Bentley CONNECT Advisor** (on page 2278)

**Automated Updates via the CONNECTION Client**

You will be notified of updates to STAAD.Pro automatically in the Bentley CONNECTION Client application. This application is installed with STAAD.Pro CONNECT Edition and runs in the Windows system tray. You can manually check for updates by opening the CONNECTION Client and selecting the **Applications** tab.

I. Using ProjectWise in STAAD.Pro

ProjectWise is an engineering project team collaboration system which is used to help teams improve quality, reduce rework, and meet project deadlines. One of the major pieces of functionality provided by ProjectWise is an Integration Server which allows data to be managed and shared across a distributed enterprise.

**Note:** This functionality requires access to a version V8i or greater of ProjectWise. For more details on ProjectWise refer to the ProjectWise client installation documentation.

**Login**

As authentication is required to access files stored on a ProjectWise repository, a login dialog allows the required details to be entered either with specific user credentials or by using the current windows login credentials.

![ProjectWise Log in dialog boxes](image)
Files that are accessed from a ProjectWise server are “Checked Out” and stored locally during the STAAD.Pro session until the file is closed and then it is returned to the server.

I. To open a STAAD input file from a ProjectWise repository

**Note:** Your most recent STAAD input files within PW repositories will be listed on the ProjectWise tab of the recent files list. You can click to select any of those files to open them.

1. On the STAAD.Pro Start page, select Open and then ProjectWise.
   You will be prompted to connect to a datasource if you have not already done so.
   The first time that a successful link to a ProjectWise server is established, a location in which checked out files are to be stored locally and additionally, where all the auxiliary data files are stored while STAAD is running, is required. Afterwards and on all future occasions, the ProjectWise open dialog presented is then presented where the repository can be navigated and filtered as defined in the ProjectWise documentation.

2. Navigate to and select the STAAD input file you want to check out and open.

   **Note:** The icons next to the STAAD filenames indicate the status of the file such as the current document, checked out to you, or locked as checked out to another user. Refer to the ProjectWise documentation for a full description of each icon.

3. Click Open.
   The STAAD input file is checked out and opened in STAAD.Pro.

**Related Links**
- *GS. Start Page* (on page 50)

I. To check in a STAAD input file to a ProjectWise repository

1. Either:
   Select the File ribbon tab and then select Close from the Backstage view menu.
   or
   Select the Close tool in the Quick Access toolbar.
   The Check In dialog opens.

2. (Optional) To increment the version of the file in the repository:
   a. Check the Create new version during Check In option.
   b. Type a Version label.
   If you choose to increment the option, you will not be able to only send updates or free the file. You must then Check In this version.

3. (Optional) To add a comment to the repository file, select the Comment tab and type your comment in the field.

4. Select the action you want to take with changes made to the file:
   To... Select...
   Check in changes made to the file since it was checked out Check In
To...  Select...
Update the server copy with changes made since the file was last checked out, but keep the file checked out  Update Server Copy
Undo the check out and prevent any changes from being made to the server copy  Free

I. To share a STAAD.Pro project in ProjectWise

To share your STAAD.Pro input files and selected files associated with that project to a ProjectWise datasource, use the following procedure.

You cannot share a project when a STAAD input file is open in the program.

Shared files are saved as STAAD.Pro Archive files (file extension .stz).

1. Select the **Share** page on the Start page.
2. Select **ProjectWise**.

3. Select the **File Name** of the STAAD input file you want to share.
   
   Click **Browse** to navigate to the file using a Windows open dialog.

4. (Optional) Select the files associated with the input file you want included in the archive.
   
   All files are selected by default.

5. Click **Add**.
   
   You are prompted to log in to a ProjectWise datasource.

   ![ProjectWise Log in dialog](image)

   - **Datasource**: ProjectWise server
   - **User Name**: Username
   - **Password**: [Redacted]
   - **Use Windows Single Sign-On for authentication**: [Optional]

6. Select your **Datasource** and then click **Log in**.

   The **Save Document As** dialog opens.
Note: Depending on your ProjectWise Explorer configuration, you may further be prompted to select a Wizard for document creation. Please refer to the ProjectWise Explorer Help for further information on this feature.

7. Select a Folder in your ProjectWise datasource and provide any optional Document data.
8. Click Save.
   The Add comment dialog opens.
9. Type a descriptive comment for the and click Add.

Your project archive is now saved to your ProjectWise datasource.

You can open this archive file from the Open > ProjectWise tab in the Start page.

Related Links
- GS. Start Page (on page 50)
- I. Archives (on page 2271)

I. Importing Models

I. To import a DXF file

To import an AutoCAD® DXF™ drawing to use for model geometry, use the following procedure.

The following table maps the imported DXF objects to model entities.
### Data Files and Interoperability

#### I. Importing Models

<table>
<thead>
<tr>
<th>DXF Object</th>
<th>STAAD Model Entity</th>
</tr>
</thead>
<tbody>
<tr>
<td>line</td>
<td>member</td>
</tr>
<tr>
<td>3Dline</td>
<td>member</td>
</tr>
<tr>
<td>3Dface</td>
<td>plate</td>
</tr>
<tr>
<td>AcDbLine</td>
<td>member</td>
</tr>
<tr>
<td>AcDb3Dpolyline</td>
<td>one or more members</td>
</tr>
<tr>
<td>AcDbFace</td>
<td>finite element plates</td>
</tr>
</tbody>
</table>

You must have a model file already open to import data.

1. Select the **File** ribbon tab.  
   The backstage view opens.
2. Select the **Import/Export** tab and then **DXF** in the **Import** group.  
   A Windows **Open** dialog opens.
3. Select the DXF file you want to import and then click **Open**.  
   The DXF Import dialog opens. This dialog is used to specify the vertical axis for data imported from a DXF file into a STAAD input file.
4. Select the **Structure Convention** to use.
   - **No Change** - Use the vertical axis as defined in the 3D DXF file
   - **Y Up** - Adds a SET Y UP command to the STAAD input file.
   - **Z Up** - Adds a SET Z UP command to the STAAD input file.
5. Click **OK**.  
   The DXF data is imported. The drawing objects are mapped to members and plates.

#### I. To import a CIS/2 file

To import a CIS/2 STEP file, use the following procedure.

The initial structural model will be created in your external 3D modeling software and exported as a CIS/2 STEP file, consisting of both analytical and physical model definitions. Limited load modeling can be done in some modeling software—such as SmartPlant® 3D—before the initial export.

**Note**: If you are importing into an empty STAAD input file, then you will be creating a new file. Otherwise, the program assumes you are updating an existing file with changes in a CIS/2 file.

1. Select the **File** ribbon tab.  
   The backstage view opens.
2. Select the **Import/Export** tab and then **CIS/2** in the **Import** group.  
   A Windows **Open** dialog opens.
3. Select the CIS/2 (*.stp) file you want to import and then click **Open**.
The Cis2Link dialog opens.

4. Either:
   for new models, click **Import Model**
   or
   for models with existing data, click **Update STAAD Model**
   The data is read and the import progress is reported in the log window.

5. (Optional) Click **Save Log File** to save this information to a text file.

6. Click **Close**.

The model data has been imported into your STAAD project.

If you need to update your STAAD.Pro model with changes made to the CIS/2 file, repeat this procedure.

**Related Links**
- *EX. CIS/2 Example Models* (on page 4310)

## I. Exporting Models

### I. To export to a DXF file

To export the model geometry to an AutoCAD DXF file, use the following procedure.

The following table maps the exported model entities to DXF objects.

<table>
<thead>
<tr>
<th>STAAD Model Entity</th>
<th>DXF Object</th>
</tr>
</thead>
<tbody>
<tr>
<td>member</td>
<td>line</td>
</tr>
<tr>
<td>element or surface</td>
<td>3Dface</td>
</tr>
<tr>
<td>solid</td>
<td>3Dface (multiple)</td>
</tr>
</tbody>
</table>

1. Select the **File** ribbon tab.
   The backstage view opens.

2. Select the **Import/Export** tab and then one of the following in the **Export** group:
   - **Option**  **Description**
     - **3D DXF**  Exports the entire model into a 3D DXF file. A Windows **Save As** dialog opens.
     - **2D DXF**  Exports a selected plane of the model into a 2D DXF file. This option opens the **Export 2D DXF** dialog.

3. If you select the 2D DXF option, you must specify the plane of the model:
   a. Select the global plane orientation to you want to use for the two dimensions.
   b. Select the node to define where this plane is cut in the model.
   c. Select the **Options** tab and then check the labels to include, if any, in the DXF drawing.
   d. Type a Text Size to use for the labels.
   e. Click **OK**.
A Windows **Save As** dialog opens.

4. Type the name of the exported file and then click **Save**.

The model geometry is exported as a drawing file.

I. To export to a CIS/2 file

To export the model geometry to a CIS/2 STEP file, use the following procedure.

1. Select the **File** ribbon tab.
   The backstage view opens.
2. Select **CIS/2** in the **Import/Export** tab.
   A Windows **Save As** dialog opens.
3. Specify the file name and location and then click **Save**.
   The **Cis2Link** dialog opens.
4. Click **Export Model**.
   The data export progress is displayed in the log window.
5. (Optional) Click **Save Log File** to save this information to a text file.
6. Click **Close**.

I. To export structure data to AutoPipe

To export support frame data to a **.NTL** file for use in AutoPipe, use the following procedure.

A piping engineer who needs to consider the steelwork as their structural supports may need to import the STAAD.Pro model into AutoPipe. A macro called **ToAutoPipePub.vbs** is available in STAAD.Pro is used to facilitate this.

1. On the **Utilities** ribbon tab, select the **User Tools Export Model to AutoPipe** tool in the **Developer** group.

   The **Export STAAD Model to AutoPipe** dialog opens.
2. Type a file path and file name for the **AutoPipe Model file** (file extension **.nt1**) to which you want to export the STAAD.Pro support frame data.

   **Tip:** Click [...] to open a Windows **Save As** dialog.
3. Click **OK**.

I. To export to a SACS input file

To export the current STAAD.Pro model to a SACS input file, use the following procedure.

You must have an input file open in STAAD.Pro.

SACS is an offshore structural analysis and design software package offered by Bentley Systems, Inc.

1. On the **Utilities** ribbon tab, select the **User Tools Export Model to SACS** tool in the **Developer** group.
The Export STAAD Model to SACS dialog opens.
2. Type a file path and file name for the SACS Model file (file extension .inp) to which you want to export the STAAD.Pro model data.

   **Tip:** Click [...] to open a Windows Save As dialog.

3. Click OK.

I. Command Line Support

You can process STAAD.Pro input files using the STAAD analysis and design engine via the Windows command line. Using this method, you can run the analysis and design engine without opening the user interface at all.

   **Tip:** It is recommended to use a batch file with the necessary commands in it in order to simplify processing and reduce the need to repeatedly type commands. You can even use the Scheduled Tasks utility in Windows to automatically run these files for you on a schedule.

I. Command Line Syntax

The following format is used when running STAAD.Pro from the command line interface.

**Syntax**

```
<path>\SProStaad\SProStaad.exe STAAD <input_file> [Options]
```

where:

- `<path>` the installation path for STAAD.Pro, which is C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\ by default.

   **Tip:** You can add “C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\ STAAD\SProStaad" to your Windows PATH environment variable to simply use the command SProStaad.

- `<input_file>` the name of the STAAD input file you want to use as input.

   **Note:** If the input file name includes spaces, you must use quote characters (") around the file name.

Expressions in square brackets (between '[' and ']') are optional, and can take one of the following values if used:

- `/s` Run silently and auto close at the end of run
- `/h` Run silently and hidden
Example
The following is a simple example which can be run from a directory containing Example.std.
C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\SProStaad.exe STAAD Example.std /s

When the analysis and design engine is complete, it will close as a result of using the 'silent' option. Otherwise, this window must be manually closed.

Batch File Example
The following can be saved as a batch file (.bat) in any text editor. This batch file will analyze and design all the US examples which ship with the product by making use of a FOR command that loops through all STAAD input files beginning with the string "Examp".

::Create a new variable with the location of the SProStaad.exe file
SET SProInst="C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\SProStaad"
::Add this to the system path variable for this session
SET PATH=%PATH%;%SProInst%
::Create a variable to the default location of the STAAD US Example files
SET USEXamps="%PUBLIC%\Documents\STAAD.Pro CONNECT Edition\Sample Models\US"
FOR %%f IN (%USEXamps%\Examp*.std) DO (ECHO %%~nf
SProStaad.exe STAAD "%%~nf.std" /h)
EXIT /B

No additional windows will open while the analysis and design engine runs as a result of using the "hidden" option.

**Note:** The use of the variables `SProInst` and `USEXamps` are not necessary. They are only used to highlight that if your installation folders differ from the default, this should be changed accordingly.

I. Copy/Paste from Spreadsheets
The Copy/Paste facility in input tables in STAAD.Pro allows different column sizes and configuration to be copied from an external spreadsheet file and then pasted into STAAD.Pro.

On any STAAD.Pro input grid table capable of handling manual input, rows and columns copied from a spreadsheet can be directly pasted into STAAD.Pro. If the columns in the spreadsheet do not match the column headers in a STAAD.Pro grid table, STAAD.Pro will ask to assign the columns in the spreadsheet to the appropriate columns in the STAAD.Pro grid table. In the figure shown below, a sample spreadsheet file containing the coordinates of some nodes is displayed. A selection of the spreadsheet's cells have been copied to the system clipboard.
This data is then selected and attempted to be copied into the STAAD.Pro Nodes table shown in the next figure. **Tip:** The entire row in the STAAD table should be selected before pasting the data. The paste function will *not* be active if only one cell is selected.

The number of columns in the first figure does not match the number of columns in the second figure. In this case, the **Select Column Mapping** dialog opens when attempting to paste the spreadsheet data.

The upper list box displays the current mapping between the spreadsheet (the **Field** column) and the STAAD.Pro table (the **Column** column). By clicking on a particular **Field**, the mapping between the spreadsheet and the STAAD.Pro table can be changed under the **Change Mapping** list box. The options listed under the **Change Mapping** list box are the available columns in the STAAD.Pro table. If a particular column coming from the spreadsheet is not relevant for the STAAD.Pro table, simply choose the option **Not Mapped** under the **Change Mapping** list box.
Tip: While the **Select Column Mapping** dialog is open, the system's ability to paste from the clipboard may be interrupted. Closing this dialog (either by clicking **OK** or **Cancel**) will restore normal clipboard functionality.
This section serves to familiarize you with the basic principles involved in the implementation of the various analysis and design facilities offered by the STAAD.Pro engine. As a general rule, the sequence in which the facilities are discussed follows the recommended sequence of their usage in the STAAD input file.

G.1 Input Generation

The STAAD input file can be created through the graphical user interface (GUI) modeling facility or using the STAAD.Pro Editor. The graphical modeling facility creates the input file through an interactive, graphics oriented procedure. In general, any plain-text editor may be utilized to edit or create the STAAD input file, but the STAAD.Pro Editor is recommended.

**Note:** Some of the automatic generation facilities of the STAAD command language will be reinterpreted by the GUI as lists of individual model elements upon editing the file using the GUI. A warning message is presented prior to this occurring. This does not result in any effective difference in the model or how it is analyzed or designed.

It is important to understand that STAAD.Pro is capable of analyzing a wide range of structures. While some parametric input features are available in the GUI, the formulation of input is the responsibility of you, the user. The program has no means of verifying that the structure input is that which was intended by the engineer.

G.2 Types of Structures

A **STRUCTURE** can be defined as an assemblage of elements. STAAD.Pro is capable of analyzing and designing structures consisting of frame, plate/shell, and solid elements. Almost any type of structure can be analyzed by STAAD.Pro.

**SPACE** A 3D framed structure with loads applied in any plane. This structure type is the most general.

**PLANE** This structure type is bound by a global X-Y coordinate system with loads in the same plane.

**TRUSS** This structure type consists of truss members which can have only axial member forces and no bending in the members.

**FLOOR** A 2D or 3D structure having no horizontal (global X or Z) movement of the structure [FX, FZ, and MY are restrained at every joint]. The floor framing (in global X-Z plane) of a building is an ideal example of this type of structure. Columns can also be modeled with the floor in a FLOOR structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a SPACE structure.
Specification of the correct structure type reduces the number of equations to be solved during the analysis. This results in a faster and more economic solution for the user. The degrees of freedom associated with frame elements of different types of structures is illustrated in the following figure.

![Figure 207: Degrees of freedom in each type of Structure](image)

**Related Links**
- TR.2 Problem Initiation and Model Title (on page 2409)

## G.3 Unit Systems

You are allowed to input data and request output in almost all commonly used engineering unit systems including “meter - kilonewton - second” (MKS), International System of Units (SI), and “feet - pound - second” (FPS). In the input file, you may change units as many times as required. Mixing and matching between length and force units from different unit systems is also allowed.

The input unit for angles (or rotations) is degrees. However, in JOINT DISPLACEMENT output, the rotations are provided in radians.

For all output, the units are clearly specified by the program.

**Related Links**
- TR.3 Unit Specification (on page 2411)

## G.4 Coordinate Systems and Structure Geometry

A structure is an assembly of individual components such as beams, columns, slabs, plates etc.. In STAAD.Pro, frame elements and plate elements may be used to model the structural components. Typically, modeling of the structure geometry consists of two steps:

A. Identification and description of joints or nodes.

B. Modeling of members or elements through specification of connectivity (incidences) between joints.
In general, the term MEMBER will be used to refer to frame elements and the term ELEMENT will be used to refer to plate/shell and solid elements. Connectivity for MEMBERs may be provided through the MEMBER INCIDENCE command while connectivity for ELEMENTs may be provided through the ELEMENT INCIDENCE command.

STAAD.Pro uses two types of coordinate systems to define the structure geometry and loading patterns. The GLOBAL coordinate system is an arbitrary coordinate system in space which is utilized to specify the overall geometry & loading pattern of the structure. A LOCAL coordinate system is associated with each member (or element) and is utilized in MEMBER END FORCE output or local load specification.

Related Links
- TR.11 Joint Coordinates Specification (on page 2425)
- TR.12 Member Incidences Specification (on page 2428)
- TR.14.2 Element Mesh Generation (on page 2435)
- TR.16.1 Listing of Entities by Specifying Groups (on page 2440)
- TR.17 Rotation of Structure Geometry (on page 2444)
- TR.26.1 Define Material (on page 2501)

G.4.1 Global Coordinate System

The following coordinate systems are available for specification of the structure geometry.

**Note:** The following figures depict the global coordinates with Y as vertical. The coordinate systems follow the similar right-hand rule when Z is vertical. Refer to SET { Y | Z } UP (on page 2418) for additional details.

**Conventional Cartesian Coordinate System**

This coordinate system is a rectangular coordinate system (X, Y, Z) which follows the orthogonal right hand rule. This coordinate system may be used to define the joint locations and loading directions.

**Cylindrical Coordinate System**

In this coordinate system, the X and Y coordinates of the conventional Cartesian system are replaced by R (radius) and Ø (angle in degrees). The Z coordinate is identical to the Z coordinate of the Cartesian system and its positive direction is determined by the right hand rule.
Reverse Cylindrical Coordinate System
This is a cylindrical type coordinate system where the R-Ø plane corresponds to the X-Z plane of the Cartesian system. The right hand rule is followed to determine the positive direction of the Y axis.

Related Links
- TR.11 Joint Coordinates Specification (on page 2425)
- TR.12 Member Incidences Specification (on page 2428)
- TR.14.2 Element Mesh Generation (on page 2435)
- TR.16.1 Listing of Entities by Specifying Groups (on page 2440)
- TR.17 Rotation of Structure Geometry (on page 2444)
- TR.26.1 Define Material (on page 2501)

G.4.2 Local Coordinate System
A local coordinate system is associated with each member. Each axis of the local orthogonal coordinate system is also based on the right hand rule.
The following figures show a beam member with start joint 'i' and end joint 'j'. The positive direction of the local x-axis is determined by joining 'i' to 'j' and projecting it in the same direction. The right hand rule may be applied to obtain the positive directions of the local y and z axes. The local y and z-axes coincide with the axes of the two principal moments of inertia. Note that the local coordinate system is always rectangular.

A wide range of cross-sectional shapes may be specified for analysis. These include rolled steel shapes, user-specified prismatic shapes etc. The following table shows local axis system(s) for these shapes.
**Table 221: Local axis system for various cross sections when global Y axis is vertical**

<table>
<thead>
<tr>
<th>Cross Section</th>
<th>Image 1</th>
<th>Image 2</th>
<th>Image 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wide Flange - ST</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Wide Flange - TB</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Wide Flange - CM</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Angle - ST</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Angle - RA</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Angle - LD (Long legs back-to-back)</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Angle - SD (Short legs back-to-back)</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Wide Flange - T</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Channel - ST</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Channel - D</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Prismatic</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>Tube - ST</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
</tbody>
</table>

*Figure 213: Labels for the local axes of a single angle as defined in AISC publications.*
Table 222: Local axis system for various cross sections when global Z axis is vertical (SET Z UP is specified).

<table>
<thead>
<tr>
<th>Section Type</th>
<th>Local Axis System</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wide Flange - ST</td>
<td><img src="image1.png" alt="Image" /></td>
</tr>
<tr>
<td>Wide Flange - TB</td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
<tr>
<td>Wide Flange - CM</td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
<tr>
<td>Angle - LD</td>
<td><img src="image4.png" alt="Image" /></td>
</tr>
<tr>
<td>Angle - SD</td>
<td><img src="image5.png" alt="Image" /></td>
</tr>
<tr>
<td>Channel - ST</td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
<tr>
<td>Wide Flange - T</td>
<td><img src="image7.png" alt="Image" /></td>
</tr>
<tr>
<td>Channel - D</td>
<td><img src="image8.png" alt="Image" /></td>
</tr>
<tr>
<td>Prismatic</td>
<td><img src="image9.png" alt="Image" /></td>
</tr>
<tr>
<td>Tube - ST</td>
<td><img src="image10.png" alt="Image" /></td>
</tr>
<tr>
<td>Angle - ST</td>
<td><img src="image11.png" alt="Image" /></td>
</tr>
<tr>
<td>Angle - RA</td>
<td><img src="image12.png" alt="Image" /></td>
</tr>
</tbody>
</table>

**Note:** The local x-axis of the above sections is going into the screen.

**Related Links**
- TR.11 Joint Coordinates Specification (on page 2425)
- TR.12 Member Incidences Specification (on page 2428)
- TR.14.2 Element Mesh Generation (on page 2435)
- TR.16.1 Listing of Entities by Specifying Groups (on page 2440)
G.4.3 Relationship Between Global and Local Coordinates

Since the input for member loads can be provided in the local and global coordinate system and the output for member-end-forces is printed in the local coordinate system, it is important to know the relationship between the local and global coordinate systems. This relationship is defined by an angle measured in the following specified way. This angle will be defined as the beta (β) angle. For offset members the beta angle/reference point specifications are based on the offset position of the local axis, not the joint positions.

**Beta Angle**

When the local x-axis is parallel to the global Vertical axis, as in the case of a column in a structure, the beta angle is the angle through which the local z-axis (or local Y for SET Z UP) has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis (global Y axis for SET Z UP).

When the local x-axis is not parallel to the global Vertical axis, the beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position of having the local z-axis (or local Y for SET Z UP) parallel to the global X-Z plane (or global X-Y plane for SET Z UP) and the local y-axis (or local z for SET Z UP) in the same positive direction as the global vertical axis. Figure 1.7 details the positions for beta equals 0 degrees or 90 degrees. When providing member loads in the local member axis, it is helpful to refer to this figure for a quick determination of the local axis system.

**Reference Point**

An alternative to providing the member orientation is to input the coordinates (or a joint number) which will be a reference point located in the member x-y plane (x-z plane for SET Z UP) but not on the axis of the member. From the location of the reference point, the program automatically calculates the orientation of the member x-y plane (x-z plane for SET Z UP).
Reference Vector

This is yet another way to specify the member orientation. In the reference point method described above, the X,Y,Z coordinates of the point are in the global axis system. In a reference vector, the X,Y,Z coordinates are specified with respect to the local axis system of the member corresponding to the BETA 0 condition.

A direction vector is created by the program. The program then calculates the Beta Angle using this vector.
Figure 215: Beta rotation of equal & unequal legged 'ST' angles

**Note:** The order of the joint numbers in the MEMBER INCIDENCES command determines the direction of the member's local x-axis.
Figure 216: Beta rotation of equal & unequal legged 'RA' angles

For unequal angles, the longer leg is shown with the dot.
Figure 217: T-shape member orientation for various Beta angles when Global-Y axis is vertical
Figure 218: T-shape member orientation for various Beta angles when Global-Z axis is vertical (that is, SET Z UP is specified)
Figure 219: Channel member orientation for various Beta angles when Global-Y axis is vertical

Related Links
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)
- Beta Angle dialog (on page 2978)
- M. To assign a member rotation angle (on page 794)
- M. To align a single angle to its flanges (on page 795)
- Reference Point dialog (on page 2979)
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)
- M. To align a member to a reference point (on page 796)
- TR.11 Joint Coordinates Specification (on page 2425)
- TR.12 Member Incidences Specification (on page 2428)
- TR.14.2 Element Mesh Generation (on page 2435)
- TR.16.1 Listing of Entities by Specifying Groups (on page 2440)
- TR.17 Rotation of Structure Geometry (on page 2444)
- TR.26.1 Define Material (on page 2501)
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)
- TR.20.8 Curved Member Specification (on page 2475)
G.5 Finite Element Information

STAAD.Pro is equipped with a plate/shell finite element, solid finite element and an entity called the surface element. The features of each is explained in the following sections.

Related Links

- **TR.11 Joint Coordinates Specification** (on page 2425)
- **TR.13 Plate and Solid Elements** (on page 2430)
- **TR.13.1 Plate and Shell Element Incidence Specification** (on page 2431)
- **TR.13.2 Solid Element Incidences Specification** (on page 2432)
- **TR.14 Plate Element Mesh Generation** (on page 2433)
- **TR.14.2 Element Mesh Generation** (on page 2435)
- **TR.21 Element/Surface Property Specification** (on page 2487)
- **TR.21.1 Element Property Specification** (on page 2487)
- **TR.24 Element Plane Stress and Ignore Inplane Rotation Specification** (on page 2498)
- **TR.32.3 Element Load Specifications** (on page 2657)
- **TR.32.3.1 Element Load Specification - Plates** (on page 2657)

G.5.1 Plate and Shell Elements

The plate (or shell) finite element is based on the hybrid element formulation. The element can be 3-noded (triangular) or 4-noded (quadrilateral). If all the four nodes of a quadrilateral element do not lie on one plane, it is advisable to model them as triangular elements. The thickness of the element may be different from one node to another.

“Surface structures” such as walls, slabs, plates and shells may be modeled using finite elements. For convenience in generation of a finer mesh of plate/shell elements within a large area, a MESH GENERATION facility is available.

You may also use the element for PLANE STRESS action only (i.e., membrane/in-plane stiffness only). The ELEMENT PLANE STRESS command should be used for this purpose.

Related Links

- **TR.21.1 Element Property Specification** (on page 2487)
- **Plate Element Property dialog** (on page 2981)
- **M. To specify plate thickness** (on page 806)
- **Plate Reference Point dialog** (on page 2981)
- **M. To align a plate to a reference point** (on page 805)
- **Plate Specs dialog** (on page 2967)
- **TR.24 Element Plane Stress and Ignore Inplane Rotation Specification** (on page 2498)
- **M. To assign plates as plane stress** (on page 808)
- **Plate Specs dialog** (on page 2967)
- **TR.24 Element Plane Stress and Ignore Inplane Rotation Specification** (on page 2498)
- **M. To assign inplane rotation behavior to plates** (on page 809)
- **Plate Specs dialog** (on page 2967)
- **TR.22.3 Element Ignore Stiffness** (on page 2491)
Geometry Modeling Considerations

1. The program automatically generates a fictitious, center node “0” (see the following figure) at the element center.

   ![Figure 220: Fictitious center node](image)

2. While assigning nodes to an element in the input data, it is essential that the nodes be specified either clockwise or counter clockwise (see the following figure). For better efficiency, similar elements should be numbered sequentially.

   ![Figure 221: Examples of correct and incorrect numbering sequences](image)
3. Element aspect ratio should not be excessive. They should be on the order of 1:1, and preferably less than 4:1.
4. Individual elements should not be distorted. Angles between two adjacent element sides should not be much larger than 90 and never larger than 180.
   a. Opposite sides cross each other
   b. Ratio of the lengths of the longest side to the shortest side exceeds eight
   c. Ratio of the sides exceeds eight
   d. Angle between two adjacent sides exceeds 120 degrees

![Figure 222: Some examples of good and bad elements in terms of the angles](image)

**Load Specification for Plate Elements**

1. Joint loads at element nodes in global directions.
2. Concentrated loads at any user specified point within the element in global or local directions.
3. Uniform pressure on element surface in global or local directions.
4. Partial uniform pressure on user specified portion of element surface in global or local directions.
5. Linearly varying pressure on element surface in local directions.
6. Temperature load due to uniform increase or decrease of temperature.
7. Temperature load due to difference in temperature between top and bottom surfaces of the element.
Theoretical Basis

The incomplete quadratic assumed stress distribution:

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} =
\begin{bmatrix}
a_1 & a_2 & a_3 & \ldots & a_{10}
\end{bmatrix}
\]

\(a_1, a_2, a_3, \ldots, a_{10} = \) constants of stress polynomials

The following quadratic stress distribution is assumed for plate bending action:

\[
\begin{bmatrix}
Q_x \\
Q_y \\
M_x \\
M_y \\
M_{xy}
\end{bmatrix}
\]

\(Q_x, Q_y, M_x, M_y, M_{xy} = \) forces and moments

The incomplete quadratic assumed stress distribution:
The distinguishing features of this finite element are:

1. Displacement compatibility between the plane stress component of one element and the plate bending component of an adjacent element which is at an angle to the first (see the following figure) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/plate elements.

![Adjacent elements at some angle](image)

2. The out of plane rotational stiffness from the plane stress portion of each element is usefully incorporated and not treated as a dummy as is usually done in most commonly available commercial software.

3. Despite the incorporation of the rotational stiffness mentioned previously, the elements satisfy the patch test absolutely.

4. These elements are available as triangles and quadrilaterals, with corner nodes only, with each node having six degrees of freedom.

5. These elements are the simplest forms of flat shell/plate elements possible with corner nodes only and six degrees of freedom per node. Yet solutions to sample problems converge rapidly to accurate answers even with a large mesh size.

6. These elements may be connected to plane/space frame members with full displacement compatibility. No additional restraints/releases are required.

7. Out of plane shear strain energy is incorporated in the formulation of the plate bending component. As a result, the elements respond to Poisson boundary conditions which are considered to be more accurate than the customary Kirchoff boundary conditions.

8. The plate bending portion can handle thick and thin plates, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.

9. The plane stress triangle behaves almost on par with the well known linear stress triangle. The triangles of most similar flat shell elements incorporate the constant stress triangle which has very slow rates of convergence. Thus the triangular shell element is very useful in problems with double curvature where the quadrilateral element may not be suitable.

10. Stress retrieval at nodes and at any point within the element.

---

**Plate Element Local Coordinate System**

1. The vector pointing from I to J is defined to be parallel to the local x-axis.
2. For triangles: the cross-product of vectors IJ and JK defines a vector parallel to the local z-axis, i.e., $z = IJ \times JK$.

For quads: the cross-product of vectors IJ and JL defines a vector parallel to the local $z$-axis, i.e., $z = IJ \times JL$.

3. The cross-product of vectors $z$ and $x$ defines a vector parallel to the local $y$-axis, i.e., $y = z \times x$.

4. The origin of the axes is at the center (average) of the four joint locations (three joint locations for a triangle).

![Figure 226: Element origin](image)

**Output of Plate Element Stresses and Moments**

**ELEMENT** stress and moment output is available at the following locations:

- **A.** Center point of the element.
- **B.** All corner nodes of the element.
- **C.** At any user specified point within the element.

Following are the items included in the **ELEMENT STRESS** output.

**Table 223: Items included in the Stress Element output**

<table>
<thead>
<tr>
<th>Title</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SQX, SQY</td>
<td>Shear stresses (Force/ unit length/ thickness)</td>
</tr>
<tr>
<td>SX, SY</td>
<td>Membrane stresses (Force/unit length/ thickness)</td>
</tr>
<tr>
<td>SXY</td>
<td>Inplane Shear Stress (Force/unit length/ thickness)</td>
</tr>
<tr>
<td>MX, MY, MXY</td>
<td>Moments per unit width (Force x Length/length)</td>
</tr>
<tr>
<td></td>
<td>(For $Mx$, the unit width is a unit distance parallel to the local $Y$ axis. For $My$, the unit width is a unit distance parallel to the local $X$ axis. $Mx$ and $My$ cause bending, while $Mxy$ causes the element to twist out-of-plane.)</td>
</tr>
<tr>
<td>SMAX, SMIN</td>
<td>Principal stresses in the plane of the element (Force/ unit area). The 3rd principal stress is 0.0</td>
</tr>
<tr>
<td>TMAX</td>
<td>Maximum 2D shear stress in the plane of the element (Force/unit area)</td>
</tr>
</tbody>
</table>
### Title Description

<table>
<thead>
<tr>
<th>VONT, VONB</th>
<th>3D Von Mises stress at the top and bottom surfaces, where:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>VM = 0.707[(SMAX - SMIN)^2 + SMAX^2 + SMIN^2] ^{1/2}</td>
</tr>
</tbody>
</table>

### Notes

1. All element stress output is in the local coordinate system. The direction and sense of the element stresses are explained in the following section.

2. To obtain element stresses at a specified point within the element, you must provide the location (local X, local Y) in the coordinate system for the element. The origin of the local coordinate system coincides with the center of the element.

3. The 2 nonzero Principal stresses at the surface (SMAX & SMIN), the maximum 2D shear stress (TMAX), the 2D orientation of the principal plane (ANGLE), the 3D Von Mises stress (VONT & VONB), and the 3D Tresca stress (TRESACAT & TRESABC) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.

4. The third principal stress is assumed to be zero at the surfaces for use in Von Mises and Tresca stress calculations. However, the TMAX and ANGLE are based only on the 2D inplane stresses (SMAX & SMIN) at the surface. The 3D maximum shear stress at the surface is not calculated but would be equal to the 3D Tresca stress divided by 2.0.
Sign Convention of Plate Element Stresses and Moments

Figure 227: Sign conventions for plate stresses and moments

Figure 228: Sign convention for plate bending

Mx is the Bending Moment on the local x face and the local x-face is the face perpendicular to the local x-axis.

My is the Bending Moment on the local y face and the local y-face is the face perpendicular to the local y-axis.
Figure 229: Stress caused by $M_x$

Figure 230: Stress caused by $M_y$
Figure 231: Torsion

Figure 232: Membrane stress $Sx$ and $Sy$
Members, plate elements, solid elements and surface elements can all be part of a single STAAD model. The MEMBER INCIDENCES input must precede the INCIDENCE input for plates, solids or surfaces. All INCIDENCES must precede other input such as properties, constants, releases, loads, etc. The selfweight of the finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.

**Plate Element Numbering**

Therefore, to save some computing time, similar elements should be numbered sequentially. The following figure shows examples of efficient and non-efficient element numbering.
However, you have to decide between adopting a numbering system which reduces the computation time versus a numbering system which increases the ease of defining the structure geometry.

Figure 235: Examples of efficient and inefficient element numbering

G.5.2 Solid Elements

Solid elements enable the solution of structural problems involving general three dimensional stresses. There is a class of problems such as stress distribution in concrete dams or soil and rock strata where finite element analysis using solid elements provides a powerful tool.

Theoretical Basis

The solid element used in STAAD.Pro is of eight-noded, isoparametric type. These elements have three translational degrees-of-freedom per node.

Figure 236: eight-noded, isoparametric solid element

By collapsing various nodes together, an eight noded solid element can be degenerated to the following forms with four to seven nodes. Joints 1, 2, and 3 must be retained as a triangle.
The stiffness matrix of the solid element is evaluated by numerical integration with eight Gauss-Legendre points (2x2x2). The integration is the reduced second order to prevent shear locking. To facilitate the numerical integration, the geometry of the element is expressed by interpolating functions using the natural coordinate system, \((r,s,t)\) of the element with its origin at the "center." The interpolating functions are shown below:

\[
X = \sum_{i=1}^{8} h_i x_i, \quad y = \sum_{i=1}^{8} h_i y_i, \quad z = \sum_{i=1}^{8} h_i z_i
\]

where

- \(x, y, z\) are the coordinates of any point in the element
- \(x_i, y_i, z_i\) are the coordinates of nodes defined in the global coordinate system, \(i=1,...,8\)

The interpolation functions, \(h_i\) are defined in the natural coordinate system, \((r,s,t)\). Each of \(r,s\), and \(t\) varies between -1 and +1. The fundamental property of the unknown interpolation functions \(h_i\) is that their values in natural coordinate system is unity at node, \(i\), and zero at all other nodes of the element. The element displacements are also interpreted the same way as the geometry. For completeness, the functions are given below:

\[
u = \sum_{i=1}^{8} h_i u_i, \quad v = \sum_{i=1}^{8} h_i v_i, \quad w = \sum_{i=1}^{8} h_i w_i
\]

where \(u, v, \) and \(w\) are displacements at any point in the element and \(u_i, v_i, w_i, i=1,8\) are corresponding nodal displacements in the coordinate system used to describe the geometry.

Three additional displacement "bubble" functions which have zero displacements at the surfaces are added in each direction for improved shear performance to form a 33x33 matrix. A modified integration is used for the bubble functions to make the results invariant with respect to element orientation (a one-point integration at the center is used).

Static condensation is used to reduce this matrix to a 24x24 matrix at the corner joints.
Local Coordinate System

The local coordinate system used in solid element is the same as the global system.

![Figure 238: Local coordinate system for a solid element](Image)

Properties and Constants

Unlike members and shell (plate) elements, no properties are required for solid elements. However, the constants such as modulus of elasticity and Poisson’s ratio are to be specified. Also, density needs to be provided if selfweight is included in any load case.

Output of Element Stresses

Element stresses may be obtained at the center and at the joints of the solid element. The items that are printed are:

- Normal Stresses: SXX, SYY and SZZ
- Shear Stresses: SXY, SYZ and SZX
- Principal stresses: S1, S2 and S3
- Von Mises stresses: $\sigma_{VM} = 0.707\sqrt{(S1 - S2)^2 + (S2 - S3)^2 + (S3 - S1)^2}$

Direction cosines: six direction cosines are printed, following the expression DC, corresponding to the first two principal stress directions.

Related Links

- M. To check for negative volume solids (on page 893)
- TR.32.3.2 Element Load Specification - Solids (on page 2660)
- TR.11 Joint Coordinates Specification (on page 2425)
- TR.13 Plate and Solid Elements (on page 2430)
- TR.13.1 Plate and Shell Element Incidence Specification (on page 2431)
- TR.13.2 Solid Element Incidences Specification (on page 2432)
- TR.14 Plate Element Mesh Generation (on page 2433)
- TR.14.2 Element Mesh Generation (on page 2435)
General Engineering Theory
G.6 Member Properties

- TR.21 Element/Surface Property Specification (on page 2487)
- TR.21.1 Element Property Specification (on page 2487)
- TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2498)
- TR.32.3 Element Load Specifications (on page 2657)
- TR.32.3.1 Element Load Specification - Plates (on page 2657)

G.5.3 Surface Elements (Deprecated)

**Note:** Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

Related Links
- TR.11 Joint Coordinates Specification (on page 2425)
- TR.13 Plate and Solid Elements (on page 2430)
- TR.13.1 Plate and Shell Element Incidence Specification (on page 2431)
- TR.13.2 Solid Element Incidences Specification (on page 2432)
- TR.14 Plate Element Mesh Generation (on page 2433)
- TR.14.2 Element Mesh Generation (on page 2435)
- TR.21 Element/Surface Property Specification (on page 2487)
- TR.21.1 Element Property Specification (on page 2487)
- TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2498)
- TR.32.3 Element Load Specifications (on page 2657)
- TR.32.3.1 Element Load Specification - Plates (on page 2657)
- TR.13.3 Surface Entities Specification (on page 2433)
- TR.21.2 Surface Property Specification (on page 2488)
- TR.26.3 Surface Constants Specification (on page 2508)
- TR.42 Print Specifications (on page 2840)
- TR.43 Stress/Force Output Printing for Surface Entities (on page 2846)
- TR.55 Shear Wall Design (on page 2862)

G.6 Member Properties

The following types of member property specifications are available in STAAD.Pro.

Shear Area for members refers to the shear stiffness effective area. Shear stiffness effective area is used to calculate shear stiffness for the member stiffness matrix.

As an example: for a rectangular cross section, the shear stiffness effective area is usually taken as 0.83 (Roark) to 0.85 (Cowper) times the cross sectional area. A shear area of less than the cross sectional area will reduce the stiffness. A typical shearing stiffness term is

\[
\frac{(12EI/L)^3}{(1+\Phi)}
\]

where

\[
\Phi = \frac{(12 EI)}{(GA_s L^2)}
\]

\[
A_s \quad \text{the shear stiffness effective area}
\]
Phi (Φ) is usually ignored in basic beam theory. STAAD will include the PHI term unless the SET SHEAR command is entered.

Shear stress effective area is a different quantity that is used to calculate shear stress and in code checking. For a rectangular cross section, the shear stress effective area is usually taken as two-thirds (0.67x) of the cross sectional area.

Shear stress in STAAD may be from one of three methods.

1. \( \frac{\text{Shear Force}}{\text{Shear stress effective area}} \)
   This is the case where STAAD computes the area based on the cross section parameters.

2. \( \frac{\text{Shear Force}}{\text{Shear stiffness effective area}} \)
   This is the case where STAAD uses the shear area entered.

3. \( \frac{V Q}{I t} \)
   In some codes and for some cross sections, STAAD uses this method.

The values that STAAD uses for shear area for shear deformation calculation can be obtained by specifying the command PRINT MEMBER PROPERTIES.

The output for this will provide this information in all circumstances: when AY and AZ are not provided, when AY and AZ are set to zero, when AY and AZ are set to very large numbers, when properties are specified using PRISMATIC, when properties are specified through a user table, when properties are specified through from the built-in-table, etc.

**Related Links**

- [TR.20 Member Property Specification](on page 2459)

**G.6.1 Prismatic Properties**

The following prismatic properties are required for analysis:

- \( AX = \text{Cross sectional area} \)
- \( IX = \text{Torsional constant} \)
- \( IY = \text{Moment of inertia about y-axis} \)
- \( IZ = \text{Moment of inertia about z-axis} \)

In addition, you may choose to specify the following properties:

- \( AY = \text{Effective shear area for shear force parallel to local y-axis} \)
- \( AZ = \text{Effective shear area for shear force parallel to local z-axis} \)
- \( YD = \text{Depth of section parallel to local y-axis} \)
- \( ZD = \text{Depth of section parallel to local z-axis} \)
For T-beams, YD, ZD, YB & ZB must be specified. These terms are:

- **YD** = Total depth of section (top fiber of flange to bottom fiber of web)
- **ZD** = Width of flange
- **YB** = Depth of stem
- **ZB** = Width of stem

For Trapezoidal beams, YD, ZD & ZB must be specified. These terms, which too are shown in the next figure are:

- **YD** = Total depth of section
- **ZD** = Width of section at top fiber
- **ZB** = Width of section at bottom fiber

The top & bottom are defined as positive side of the local Z axis, and negative side of the local Z axis respectively.

STAAD.Pro automatically considers the additional deflection of members due to pure shear (in addition to deflection due to ordinary bending theory). To ignore the shear deflection, enter a SET SHEAR command before the joint coordinates. This will bring results close to textbook results.

The depths in the two major directions (YD and ZD) are used in the program to calculate the section moduli. These are needed only to calculate member stresses or to perform concrete design. You can omit the YD & ZD values if stresses or design of these members are of no interest. The default value is 253.75 mm (9.99 inches) for YD and ZD. All the prismatic properties are input in the local member coordinates.

To define a concrete member, you must not provide AX, but instead, provide YD and ZD for a rectangular section and just YD for a circular section. If no moment of inertia or shear areas are provided, the program will automatically calculate these from YD and ZD.

The following table is provided to assist you in specifying the necessary section values. It lists, by structural type, the required section properties for any analysis.

**Note:** For the PLANE or FLOOR type analyses, the choice of the required moment of inertia depends upon the beta angle. If BETA equals zero, the required property is IZ.

### Table 224: Required Section Properties

<table>
<thead>
<tr>
<th>Structure Type</th>
<th>Required Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRUSS</td>
<td>AX</td>
</tr>
<tr>
<td>PLANE</td>
<td>AX, IZ, or IY</td>
</tr>
</tbody>
</table>
### Structure Type

<table>
<thead>
<tr>
<th>Structure Type</th>
<th>Required Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLOOR</td>
<td>IX, IZ or IY</td>
</tr>
<tr>
<td>SPACE</td>
<td>AX, IX, IY, IZ</td>
</tr>
</tbody>
</table>

### Related Links

- [Property dialog](on page 2970)
- [TR.20.2 Prismatic Property Specification](on page 2465)
- [TR.20.2.1 Prismatic Tapered Tube Property Specification](on page 2467)
- [M. To assign a prismatic section](on page 733)
- [TR.20.2 Prismatic Property Specification](on page 2465)

### G.6.2 Built-In Steel Section Libraries

This feature of the program allows you to specify section names of standard steel shapes manufactured in different countries.

Since the shear areas of the sections are built into the tables, shear deformation is always considered for these sections.

### Related Links

- [Properties - Whole Structure dialog](on page 2968)
- [TR.20 Member Property Specification](on page 2459)
- [Section Profile Tables dialog](on page 2969)
- [TR.20.1 Assigning Properties from Steel Tables](on page 2461)
- [M. To add a new table section property](on page 730)
- [D1.A.4 Built-in Steel Section Library](on page 1368)
- [TR.20.1 Assigning Properties from Steel Tables](on page 2461)

### G.6.3 User-Provided Steel Table

You can provide a customized steel table with designated names and proper corresponding properties. The program can then find member properties from those tables. Member selection may also be performed with the program selecting members from the provided tables only.

These tables can be provided as a part of a STAAD.Pro input or as separately created files from which the program can read the properties. If you do not use standard rolled shapes or only use a limited number of specific shapes, you can create permanent member property files. Analysis and design can be limited to the sections in these files.

### Related Links

- [TR.19 User Steel Table Specification](on page 2446)
- [TR.20.4 Property Specification from User Provided Table](on page 2469)
- [Create User Provided Table dialog](on page 2974)
- [User Provided Table dialog](on page 2973)
- [M. To create a general section](on page 735)
- [TR.19 User Steel Table Specification](on page 2446)
G.6.4 Tapered Sections

Properties of tapered I-sections may be provided through MEMBER PROPERTY specifications. Given key section dimensions, the program is capable of calculating cross-sectional properties which are subsequently used in analysis.

Tapered I-sections have constant flange dimensions and a linearly varying web depth along the length of the member.

Related Links

- TR.20.3 Tapered Member Specification (on page 2468)
- Property: Tapered I dialog (on page 2972)
- M. To assign a tapered I section (on page 734)
- TR.20.3 Tapered Member Specification (on page 2468)

G.6.5 Assign Command

If you want to avoid the trouble of defining a specific section name but rather leave it to the program to assign a section name, the ASSIGN command may be used. The section types that may be assigned include BEAM, COLUMN, CHANNEL, ANGLE and DOUBLE ANGLE.

When the keyword BEAM is specified, the program will assign an I-shaped beam section (Wide Flange for AISC, UB section for British).

For the keyword COLUMN also, the program will assign an I-shaped beam section (Wide Flange for AISC, UC section for British).

If steel design-member selection is requested, a similar type section will be selected.

Related Links

- TR.20.5 Assign Profile Specification (on page 2470)

G.6.6 Steel Joist and Joist Girders

STAAD.Pro includes facilities for specifying steel joists and joist girders. The basis for this implementation is the information contained in the 1994 publication of the American Steel Joist Institute called Standard Specifications, Load Tables, and Weight Tables for Steel Joists and Joist Girders, Fortieth edition. The following are the salient features of the implementation.

Member properties can be assigned by specifying a joist designation contained in tables supplied with the program. The following joists and joist girder types have been implemented:

- Open web steel joists – K series and KCS joists
- Longspan steel joists – LH series
- Deep Longspan steel joists – DLH series
- Joist Girders – G series

The designation for the G series Joist Girders is as shown in the Steel Joist Institute publication. STAAD.Pro incorporates the span length also in the name, as shown in the next figure.
Theoretical basis for modeling the joist

Steel joists are prefabricated, welded steel trusses used at closely spaced intervals to support floor or roof decking. Thus, from an analysis standpoint, a joist is not a single member in the same sense as beams and columns of portal frames that one is familiar with. Instead, it is a truss assembly of members. In general, individual manufacturers of the joists decide on the cross section details of the members used for the top and bottom chords, and webs of the joists. So, joist tables rarely contain any information on the cross-section properties of the individual components of a joist girder. The manufacturer's responsibility is to guarantee that, no matter what the cross section details of the members are, the joist simply has to ensure that it provides the capacity corresponding to its rating.

The absence of the section details makes it difficult to incorporate the true truss configuration of the joist in the analysis model of the overall structure. In STAAD, selfweight and any other member load applied on the joist is transferred to its end nodes through simply supported action. Also, in STAAD, the joist makes no contribution to the stiffness of the overall structure.

As a result of the above assumption, the following points must be noted with respect to modeling joists:

1. The entire joist is represented in the STAAD input file by a single member. Graphically it will be drawn using a single line.
2. After creating the member, the properties should be assigned from the joist database.
3. The 3D Rendering feature of the program will display those members using a representative Warren type truss.
4. The intermediate span-point displacements of the joist cannot be determined.
Assigning the joists

The procedure for assigning the joists is explained in the Graphical User Interface manual. The STAAD joists database includes the weight per length of the joists. So, for selfweight computations in the model, the weight of the joist is automatically considered.

G.6.7 Composite Beams and Composite Decks

There are two methods in STAAD.Pro for specifying composite beams. Composite beams are members whose property is comprised of an I-shaped steel cross section (like an American W shape) with a concrete slab on top. The steel section and concrete slab act monolithically. The two methods are:

1. **The EXPLICIT definition method** (on page 2461) – In this method, the member geometry is first defined as a line. It is then assigned a property from the steel database, with the help of the “CM” attribute. Additional parameters like CT (thickness of the slab), FC (concrete strength), CW (effective width of slab), CD (concrete density), etc., some optional and some mandatory, are also provided.

   Hence, the responsibility of determining the attributes of the composite member, like concrete slab width, lies upon you, the user. If you wish to obtain a design, additional terms like rib height, rib width, etc. must also be separately assigned with the aid of design parameters. Hence, some amount of effort is involved in gathering all the data and assigning them.

2. **The composite deck generation method** (on page 2473) – The laboriousness of the previous procedure can be alleviated to some extent by using the program’s composite deck definition facilities. The program then internally converts the deck into individual composite members (calculating attributes like effective width in the process) during the analysis and design phase. The deck is defined best using the graphical tools of the program since a database of deck data from different manufacturers is accessible from easy-to-use dialogs. Since all the members which make up the deck are identified as part of a single object, load assignment and alterations to the deck can be done to just the deck object, and not the individual members of the deck.

**Related Links**

- [Composite Deck dialog](on page 2916)
G.6.8 Curved Members

Members can be defined as being curved.
Tapered sections are not permitted. The cross-section should be uniform throughout the length.
The design of curved members is not supported.

Related Links
- TR.20.8 Curved Member Specification (on page 2475)
- M. To add a curved beam (on page 646)
- TR.20.8 Curved Member Specification (on page 2475)
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)

G.7 Member and Element Release

STAAD.Pro allows releases for members and plate elements.

One or both ends of a member or element can be released in any combination of the six degrees of freedom. Members and elements are assumed to be rigidly framed into joints in accordance with the structural type specified. When this full rigidity is not applicable, individual force components at either end of the member can be set to zero with member release statements. By specifying release components, individual degrees of freedom are removed from the analysis. Release components are given in the local coordinate system for each member. Partial moment releases are also allowed.

The translational degrees of freedom are denoted by \( u_1, u_2, u_3 \) and the rotational degrees of freedom are denoted by \( u_4, u_5 \) & \( u_6 \).

![Figure 242: Degrees of Freedom](image)

Note: A specific member can have either end releases (see G.7 Member and Element Release (on page 2329)) or axial-only specifications (see G.8 Axial-Only Specifications (on page 2330)) properties assigned. The last
specification for a given member will be used. In other words, a MEMBER RELEASE should not be applied on a member which is declared TRUSS, TENSION ONLY, COMPRESSION ONLY, or CABLE.

Related Links

- Plate Specs dialog (on page 2967)
- TR.22.2 Element Release Specification (on page 2490)
- M. To assign plate corner release (on page 807)
- TR.22.1 Member Release Specification (on page 2488)
- Member Specification dialog (on page 2962)
- M. To assign member end release (on page 799)
- TR.22 Member and Element Releases (on page 2488)
- TR.22.1 Member Release Specification (on page 2488)
- TR.22.2 Element Release Specification (on page 2490)
- TR.22.3 Element Ignore Stiffness (on page 2491)

G.8 Axial-Only Specifications

STAAD.Pro has several member specifications which allow you to designate axial-only behavior for members.

Note: A specific member can have either end releases (see G.7 Member and Element Release (on page 2329)) or axial-only specifications (see G.8 Axial-Only Specifications (on page 2330)) properties assigned. The last specification for a given member will be used. In other words, a MEMBER RELEASE should not be applied on a member which is declared TRUSS, TENSION ONLY, COMPRESSION ONLY, or CABLE.

G.8.1 Truss and Tension- or Compression-Only Members

For analyses which involve members that carry axial loads only (i.e., truss members) there are two methods for specifying this condition. When all the members in the structure are truss members, the type of structure is declared as TRUSS whereas, when only some of the members are truss members (e.g., bracing members in a building), the MEMBER TRUSS command can be used where those members will be identified separately.

In STAAD.Pro, the MEMBER TENSION or MEMBER COMPRESSION command can be used to limit the axial load type the member may carry.

Related Links

- TR.23.1 Member Truss Specification (on page 2492)
- TR.23.3 Member Tension/Compression Specification (on page 2495)
- Member Specification dialog (on page 2962)
- M. To assign axial action members (on page 798)
- TR.23.1 Member Truss Specification (on page 2492)
- TR.23.3 Member Tension/Compression Specification (on page 2495)

G.8.2 Cable Members

STAAD.Pro supports the following types of analysis for cable members:
G.8.2.1 Linearized Cable Members

Cable members may be specified by using the MEMBER CABLE command. While specifying cable members, the initial tension in the cable must be provided. The following paragraph explains how cable stiffness is calculated.

The increase in length of a loaded cable is a combination of two effects. The first component is the elastic stretch, and is governed by the familiar spring relationship:

\[ F = Kx \]

where \( K_{\text{elastic}} = \frac{EA}{L} \)

The second component of the lengthening is due to a change in geometry (as a cable is pulled taut, sag is reduced). This relationship can be described by

\[ F = Kx \]

but here,

\[ K_{\text{sag}} = \frac{12T^3}{w^2L^3\left(\frac{1}{\cos^2\alpha}\right)} \]

where

\( w \) = weight per unit length of cable
\( T \) = tension in cable
\( \alpha \) = angle that the axis of the cable makes with a horizontal plane (= 0, cable is horizontal; = 90, cable is vertical)

Therefore, the "stiffness" of a cable depends on the initial installed tension (or sag). These two effects may be combined as follows:

\[ K_{\text{comb}} = \frac{1}{1/K_{\text{sag}} + 1/K_{\text{elastic}}} = \frac{(EA/L)}{1 + \frac{w^2L^2E\cos^2\alpha}{12T^3}} \]

**Note:** When \( T = \infty \) (infinity), \( K_{\text{comb}} = EA/L \) and that when \( T = 0 \), \( K_{\text{comb}} = 0 \). It should also be noted that as the tension increases (sag decreases) the combined stiffness approaches that of the pure elastic situation.

The following points need to be considered when using the cable member in STAAD:

1. The linear cable member is only a truss member whose properties accommodate the sag factor and initial tension. The behavior of the cable member is identical to that of the truss member. That is, it can carry axial loads only. As a result, the fundamental rules involved in modeling truss members have to be followed when modeling cable members.

   For example, when two cable members meet at a common joint, if there is not a support or a 3rd member connected to that joint, it is a point of potential instability.

2. Due to the reasons specified in 1) above, applying a transverse load on a cable member is not advisable. The load will be converted to two concentrated loads at the two ends of the cable and the true deflection pattern of the cable will never be realized.

3. A tension only cable member offers no resistance to a compressive force applied at its ends. When the end joints of the member are subjected to a compressive force, they “give in” thereby causing the cable to sag. Under these circumstances, the cable member has zero stiffness and this situation has to be accounted for in the stiffness matrix and the displacements have to be recalculated. But in STAAD.Pro, merely declaring the
member to be a cable member does not guarantee that this behavior will be accounted for. It is also important that you declare the member to be a tension only member by using the MEMBER TENSION command, after the CABLE command. This will ensure that the program will test the nature of the force in the member after the analysis and if it is compressive, the member is switched off and the stiffness matrix recalculated.

4. Due to potential instability problems explained in item 1 above, you should also avoid modeling a catenary by breaking it down into a number of straight line segments. The cable member in STAAD.Pro cannot be used to simulate the behavior of a catenary.

By catenary, we are referring to those structural components which have a curved profile and develop axial forces due to their self weight. This behavior is in reality a nonlinear behavior where the axial force is caused because of either a change in the profile of the member or induced by large displacements, neither of which are valid assumptions in an elastic analysis. A typical example of a catenary is the main U shaped cable used in suspension bridges.

5. The increase of stiffness of the cable as the tension in it increases under applied loading is updated after each iteration if the cable members are also declared to be MEMBER TENSION. However, iteration stops when all tension members are in tension or slack; not when the cable tension converges.

Related Links
- TR.23.2 Member Cable Specification (on page 2493)
- TR.37.3 Nonlinear Cable Analysis (on page 2800)
- TR.23.1 Member Truss Specification (on page 2492)

G.8.2.2 Nonlinear Cable and Truss Members

Cable members for the Nonlinear Cable Analysis may be specified by using the MEMBER CABLE command. While specifying cable members, the initial tension in the cable or the unstressed length of the cable must be provided. You should ensure that all cables will be in sufficient tension for all load cases to converge. Use selfweight in every load case and temperature if appropriate; i.e., don’t enter component cases (e.g., wind only).

The nonlinear cable may have large motions and the sag is checked on every load step and every equilibrium iteration.

In addition there is a nonlinear truss which is specified in the Member Truss command. The nonlinear truss is simply any truss with pretension specified. It is essentially the same as a cable without sag. This member takes compression. If all cables are taut for all load cases, then the nonlinear truss may be used to simulate cables. The reason for using this substitution is that the truss solution is more reliable.

Points 1, 2, and 4 in the previous section will not apply to nonlinear cable analysis if sufficient pretension is applied, so joints may be entered along the shape of a cable (in some cases a stabilizing stiffness may be required and entered for the first loadstep). Point 3 above (on page 2331): The Member Tension command is unnecessary and ignored for the nonlinear cable analysis. Point 5 above (on page 2331): The cable tensions are iterated to convergence in the nonlinear cable analysis.

Related Links
- TR.23.2 Member Cable Specification (on page 2493)
- TR.37.3 Nonlinear Cable Analysis (on page 2800)
- TR.23.1 Member Truss Specification (on page 2492)

G.8.2.3 Nonlinear Cable Members for Advanced Cable Analysis

Cable members used for a Advanced Cable Analysis are specified by using the MEMBER CABLE command.
When specifying cable members, either the initial tension in the cable or the unstressed length of the cable must first be provided. The initial tension is sufficient for keeping the cable in tension because the catenary theory behind the element formulation will prevent the cable from behaving in compression. However, a larger initial tension (or smaller unstressed length) increases the numerical stability.

A nonlinear cable may have large deformation. The force equilibrium at cable ends is checked on every load step and on every equilibrium iteration.

Following considerations exist in Advanced Cable Analysis:

1. The nonlinear cable element provides stiffness and resistance forces to only three translational degrees (i.e., FX, FY, and FZ). It is not able to carry any moments. So, when two nonlinear cable elements meet at a common joint—and if there isn’t a support or a 3rd member connected to the joint—it is a point of potential instability. The program introduces a very weak spring to overcome this problem.

2. Due to the reason as described in item 1, applying any moment to a cable element is not advisable.

3. The cable is not able to carry any compression forces. So, when the load tends to cause compression in a cable—and if there are no other members connected to the cable and the cable can deform freely—there will be some numerical instability.

4. Due to the instability problem explained in item 1, subdividing a cable member into several smaller cable elements should be done only when it is necessary. An example of one such necessary case may be when there is force load applied in the middle of the cable member. For this case, the cable member has to be broken so that the forces can be applied as joint loads at cable nodes.

5. The increase of stiffness of the cable (by increasing the initial tension or decreasing unstressed length) is always beneficial for numerical stability. The iteration will stop only when the force equilibrium is reached.

6. In advanced cable analysis, the self weight of a cable member is initially considered to obtain the initial configuration of cable members under self weight. Any additional weight required along with self weight can be included using FWY parameter in the MEMBER CABLE command.

Note: This analysis feature can be used only when Advanced Analysis license is active.

Related Links

- TR.23.2 Member Cable Specification (on page 2493)
- TR.37.3 Nonlinear Cable Analysis (on page 2800)

G.9 Connection Tags

Connection tags provide a means for assigning both connection type data and connection capacities, which can be used for checking in STAAD.Pro.

Note: This feature is available in STAAD.Pro V8i (SELECTseries 4), release 20.07.09.21 and higher.

Connection tags consist of two pieces of data:

1. A Connection Tags XML file, which contains the connection categories, tag names, and member end releases for the connection tag. Connection capacities are also specified for each combination of member and connecting member which may utilize a connection tag. Refer to "Connection Tags XML File Schema" in the STAAD User Interface help for additional information on the required structure of this XML file.

2. Assignments of connection tags to members are stored in the STAAD input file. Though this is done within the DEFINE MEMBER ATTRIBUTE command, it is strongly recommended that the user interface features be used to make connection tag assignments as these must utilize only the connection categories and tag names.
Some members of a structure may not be concurrent with the incident joints thereby creating offsets. This offset distance is specified in terms of global or local coordinate system (i.e., X, Y and Z distances from the incident joint). Secondary forces induced, due to this offset connection, are taken into account in analyzing the structure and also to calculate the individual member forces. The new offset centroid of the member can be at the start or end incidences and the new working point will also be the new start or end of the member. Therefore, any reference from the start or end of that member will always be from the new offset points.

Related Links
- TR.25 Member Offset Specification (on page 2499)
- Member Specification dialog (on page 2962)
- M. To assign member end offsets (on page 800)
- TR.25 Member Offset Specification (on page 2499)

G.12 Material Properties

STAAD.Pro allows you to define the properties of materials by two different methods. By assigning individual constants to members, elements, or solids or by creating a material definition and then assigning this.

Material Constants

The material constants are:

- modulus of elasticity (E),
- weight density (DEN),
- Poisson's ratio (POISS),
- coefficient of thermal expansion (ALPHA),
- composite damping ratio,
- and beta angle (BETA) or coordinates for any reference (REF) point.

The modulus of elasticity, E, value for members must be provided or the analysis will not be performed. Weight density (DEN) is used only when selfweight of the structure is to be taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (commonly known as G) by the formula,

\[ G = 0.5 \cdot \frac{E}{1 + \text{POISS}} \]

If Poisson's ratio is not provided, STAAD.Pro will assume a value for this quantity based on the value of E.

**Note:** Poisson's Ratio must always be defined after the Modulus of Elasticity for a given member/element.
The coefficient of thermal expansion (\textit{ALPHA}) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and \textit{ALPHA} has to be the same.

The composite damping ratio is used to compute the damping ratio for each mode in a dynamic solution. This is only useful if there are several materials with different damping ratios.

\textit{BETA} angle and \textit{REF}erence point are discussed in \textit{G.4.3 Relationship Between Global and Local Coordinates} (on page 2301) and are input as part of the member constants.

Material Definitions

Alternately, you may define the constants of a material in a material definition. This can include the analytical and design properties for the material. These material definitions are then assigned to the members, elements, and solids.

\textbf{Note:} A \textit{BETA} is a geometric property and must be assigned separately from the material definition.

\textbf{Related Links}

- \textit{TR.26 Specifying and Assigning Material Constants} (on page 2501)

\section*{G.13 Supports}

STAAD.Pro allows specifications of supports that are parallel as well as inclined to the global axes.

Supports are specified as \textit{PINNED}, \textit{FIXED}, or \textit{FIXED} with different releases. A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement.

The restraints of a fixed support can also be released in any desired direction as specified.

Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is defined as the force to displace a support joint one length unit in the specified global direction. Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction.

For static analysis, Multi-linear spring supports can be used to model the varying, non-linear resistance of a support (e.g., soil).

The Support command is also used to specify joints and directions where support displacements will be enforced.

\textbf{Related Links}

- \textit{M. To assign a fixed or pinned support} (on page 812)
- \textit{Create Support dialog} (on page 2983)
- \textit{Supports - Whole Structure dialog} (on page 2982)
- \textit{TR.27.1 Global Support Specification} (on page 2514)
- \textit{M. To assign a foundation support} (on page 818)
- \textit{TR.27.3 Automatic Spring Support Generator for Foundations} (on page 2517)
- \textit{Create Support dialog} (on page 2983)
- \textit{M. To assign an enforced support} (on page 813)
- \textit{TR.27.1 Global Support Specification} (on page 2514)
G.9 Tension- and Compression- Only Springs

In STAAD.Pro, the SPRING TENSION or SPRING COMPRESSION command can be used to limit the load direction the support spring may carry. The analysis will be performed accordingly.

Related Links
- TR.27.5 Spring Tension/Compression Specification (on page 2522)
- TR.23.1 Member Truss Specification (on page 2492)

G.14 Rigid Diaphragms

STAAD.Pro provides the following methods of modeling rigid diaphragms for structures.

Master/Slave Joints

The master/slave option is provided to enable you to model rigid links in the structural system. This facility can be used to model special structural elements like a rigid floor diaphragm. Several slave joints may be provided which will be assigned same displacements as the master joint. You are also allowed the flexibility to choose the specific degrees of freedom for which the displacement constraints will be imposed on the slaved joints. If all degrees of freedom (Fx, Fy, Fz, Mx, My, and Mz) are provided as constraints, the joints will be assumed to be rigidly connected.

Floor Diaphragms

Alternately, you can define a floor diaphragm without the need to specify a master joint. You can specify a floor by a height range or by a joint list using this method. The program will internally calculate the center of mass of the floor and generate an analytical node at that location.

Related Links
- TR.28.1 Master/Slave Specification (on page 2525)
- Node Specification dialog (on page 2960)
- M. To assign a rigid link between nodes (on page 810)
- TR.28.1 Master/Slave Specification (on page 2525)
G.15 Loads

Loads in a structure can be directly specified for joints, members, and elements. STAAD.Pro can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self-weight can also be applied in any desired direction.

G.15.1 Joint Loads

Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint.

Related Links
- TR.32.1 Joint Load Specification (on page 2651)
- Nodal Load tab (on page 3008)
- M. To add a nodal load (on page 824)
- G.17.3 Dynamic Analysis (on page 2362)
- TR.32.1 Joint Load Specification (on page 2651)
- TR.32.10 Dynamic Loading Specification (on page 2686)

G.15.2 Member Load

Three types of member loads may be applied directly to a member of a structure: uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal).

Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads.

Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length. See G.4.1 Global Coordinate System (on page 2296) to find the relation of the member to the global coordinate systems for specifying member loads. Positive forces act in the positive coordinate directions, local or global, as the case may be.

Uniform moment may not be applied to tapered members. Only uniform load over the entire length is available for curved members.
Figure 243: Member Load Configurations for A) linear loads, B) concentrated loads, C) linear loads, D) trapezoidal load, E) triangular (linear) loads, and F) uniform load

Related Links
- TR.32.2 Member Load Specification (on page 2653)
- Member Load tab (on page 3009)
- M. To add a concentrated force or moment on members (on page 826)
- M. To add a uniform load to members (on page 826)
- M. To add a linear varying load to members (on page 827)
- M. To add a hydrostatic load to objects (on page 833)
- TR.32.2 Member Load Specification (on page 2653)

G.15.3 Area, One-way, and Floor Loads

Often a floor is subjected to a uniform pressure. It could require a lot of work to calculate the equivalent member load for individual members in that floor. However, with the AREA, ONEWAY or FLOOR LOAD facilities, you can specify the pressure (load per unit square area). The program will calculate the tributary area for these members and calculate the appropriate member loads. The Area Load and Oneway load are used for one way distribution and the Floor Load is used for two way distribution.

The following assumptions are made while transferring the area/floor load to member load:

a. The member load is assumed to be a linearly varying load for which the start and the end values may be of different magnitude.
b. Tributary area of a member with an area load is calculated based on half the spacing to the nearest approximately parallel members on both sides. If the spacing is more than or equal to the length of the member, the area load will be ignored. Oneway load does not have this limitation.

c. These loading types should not be specified on members declared as MEMBER CABLE, MEMBER TRUSS, MEMBER TENSION, MEMBER COMPRESSION, or CURVED.

**Note:** Floor Loads and One-way Loads can be reduced when included in a load case defined as "Reducible" according to the UBC/IBC rules.

An example:

Member 1 will have a linear load of 0.3 at one end and 0.2 at the other end. Members 2 and 4 will have a uniform load of 0.5 over the full length. Member 3 will have a linear load of 0.45 and 0.55 at respective ends. Member 5 will have a uniform load of 0.25. The rest of the members, 6 through 13, will have no contributory area load since the nearest parallel members are more than each of the member lengths apart. However, the reactions from the members to the girder will be considered.

Only member loads are generated from the Area, Oneway, and Floor load input. Thus, load types specific to plates, solids or surface are not generated. That is because, the basic assumption is that, a floor load or area load is used in situations where the basic entity (plate, solid or surface) which acts as the medium for application of that load, is not part of the structural model.

The Oneway load is intended to be used in areas with relatively large aspect ratios between adjacent sides. It should not be used on members with square tributary areas unless the TOWARDS option is used, which then specifies to which of the two equal directions the load should be applied. Otherwise, the Floor Load type should be used.
Note: Failure to specify a TOWARD side on a Oneway load applied to a square tributary area will likely result in lost load or unintended load path changes.

Related Links
- TR.32.4.1 Area Load Specification (on page 2665)
- Area Load tab (on page 3012)
- M. To add an area load (on page 830)
- Floor Load tab (on page 3012)
- TR.32.4.2 One-way Load Specification (on page 2666)
- TR.32.4.3 Floor Load Specification (on page 2672)
- M. To add a floor load or one-way load (on page 831)
- TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664)
- TR.32.4.1 Area Load Specification (on page 2665)
- TR.32.4.2 One-way Load Specification (on page 2666)
- TR.32.4.3 Floor Load Specification (on page 2672)

G.15.4 Fixed End Member Load

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial; shear y; shear z; torsion; moment y, and moment z.

Related Links
- TR.32.7 Fixed-End Load Specification (on page 2683)
- Member Load tab (on page 3009)
- M. To add fixed end member loads (on page 829)
- TR.32.7 Fixed-End Load Specification (on page 2683)

G.15.5 Prestress and Poststress Member Load

Members in a structure may be subjected to prestress load for which the load distribution in the structure may be investigated. The prestressing load in a member may be applied axially or eccentrically. The eccentricities can be provided at the start joint, at the middle, and at the end joint. These eccentricities are only in the local y-axis. A positive eccentricity will be in the positive local y-direction. Since eccentricities are only provided in the local y-axis, care should be taken when providing prismatic properties or in specifying the correct BETA angle when rotating the member coordinates, if necessary. Two types of prestress load specification are available; PRESTRESS, where the effects of the load are transmitted to the rest of the structure, and POSTSTRESS, where the effects of the load are experienced exclusively by the members on which it is applied.

1. The cable is assumed to have a generalized parabolic profile. The equation of the parabola is assumed to be

\[ y = ax^2 + bx + c \]

where

- \( a = \frac{1}{L^2} (2es - 4em + 2ee) \)
- \( b = \frac{1}{L} (4em - ee - 3es) \)
- \( c = es \)
- \( es \) = eccentricity of the cable at the start of the member (in local y-axis)
- \( em \) = eccentricity of the cable at the middle of the member (in local y-axis)
- \( ee \) = eccentricity of the cable at the end of the member (in local y-axis)
2. The angle of inclination of the cable with respect to the local x-axis (a straight line joining the start and end joints of the member) at the start and end points is small which gives rise to the assumption that 
\[ \sin \theta = \theta = \frac{dy}{dx} \]

Hence, if the axial force in the cable is \( P \), the vertical component of the force at the ends is \( P \left( \frac{dy}{dx} \right) \) and the horizontal component of the cable force is,
\[ P \left[ 1 - \left( \frac{dy}{dx} \right)^2 \right]^{1/2} \]

Users are advised to ensure that their cable profile meets this requirement. An angle under 5 degrees is recommended.

3. The member is analyzed for the prestressing / poststressing effects using the equivalent load method. This method is well documented in most reputed books on Analysis and Design of Prestressed concrete. The magnitude of the uniformly distributed load is calculated as
\[ udl = 8 \cdot \frac{Pe}{L^2} \]

where
\[ P = \text{Axial force in the cable} \]
\[ e = \frac{(es + ee)}{2 \cdot em} \]
\[ L = \text{Length of the member} \]

4. The force in the cable is assumed to be same throughout the member length. No reduction is made in the cable forces to account for friction or other losses.

5. The term MEMBER PRESTRESS as used in STAAD.Pro signifies the following condition. The structure is constructed first. Then, the prestressing force is applied on the relevant members. As a result, the members deform and depending on their end conditions, forces are transmitted to other members in the structure. In other words, "PRE" refers to the time of placement of the member in the structure relative to the time of stressing.

6. The term MEMBER POSTSTRESS as used in STAAD.Pro signifies the following condition. The members on which such load is applied are first cast in the factory. Following this, the prestressing force is applied on them. Meanwhile, the rest of the structure is constructed at the construction site. Then, the prestressed members are brought and placed in position on the partially built structure. Due to this sequence, the effects of prestressing are "experienced" by only the prestressed members and not transmitted to the rest of the structure. In other words, "POST" refers to the time of placement of the member in the structure relative to the time of stressing.

7. As may be evident from Item (6) above, it is not possible to compute the displacements of the ends of the POSTSTRESSED members for the effects of poststressing, and hence are assumed to be zero. As a result, displacements of intermediate sections (See SECTION DISPLACEMENT command) are measured relative to the straight line joining the start and end joints of the members as defined by their initial JOINT COORDINATES.

Related Links
- TR.32.5 Prestress Load Specification (on page 2678)
- Member Load tab (on page 3009)
- M. To add a prestress or post-tension load to members (on page 828)
- TR.32.5 Prestress Load Specification (on page 2678)

G.15.6 Temperature and Strain Load

Uniform temperature difference throughout members and elements may be specified. Temperature differences across both faces of members and through the thickness of plates may also be specified (uniform temperature...
only for solids). The program calculates the axial strain (elongation and shrinkage) due to the temperature difference for members. From this it calculates the induced forces in the member and the analysis is done accordingly. The strain intervals of elongation and shrinkage can be input directly.

Related Links
- TR.32.6 Temperature Load Specification for Members, Plates, and Solids (on page 2682)

G.15.7 Support Displacement Loads

Static Loads can be applied to the structure in terms of the displacement of the supports. Displacement can be translational or rotational. Translational displacements are provided in the specified length while the rotational displacements are always in degrees. Note that displacements can be specified only in directions in which the support has an “enforced” specification in the Support command.

Related Links
- TR.32.8 Support Joint Displacement Specification (on page 2683)
- Nodal Load tab (on page 3008)
- M. To add a support displacement (on page 825)
- TR.32.8 Support Joint Displacement Specification (on page 2683)

G.15.8 Loading on Elements

On Plate/Shell elements, the types of loading that are permissible are:

1. Pressure loading which consists of loads which act perpendicular to the surface of the element. The pressure loads can be of uniform intensity or trapezoidally varying intensity over a small portion or over the entire surface of the element.
2. Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.
3. Temperature loads which may be constant throughout the plate element (causing only elongation / shortening) or may vary across the depth of a plate element causing bending on the plate element. The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.
4. The self-weight of the elements can be applied using the SELFWEIGHT loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.

On Solid elements, the loading types available are:

1. The self-weight of the solid elements can be applied using the SELFWEIGHT loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.
2. Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.
3. Temperature loads which may be constant throughout the solid elements (causing only elongation / shortening). The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.
4. Pressure on the faces of solids.

Only translational stiffness is supported in solid elements. Thus, at joints where there are only solid elements, moments may not be applied. For efficiency, rotational supports should be used at these joints.

Related Links
- Plate Loads tab (on page 3013)
- TR.32.3.1 Element Load Specification - Plates (on page 2657)
- M. To add pressure load on a plate (on page 829)
G.16 Load Generator

Load generation is the process of taking a load causing unit such as wind pressure, ground movement or a truck on a bridge, and converting it to a form such as member load or a joint load which can be then be used in the analysis.

For seismic loads, a static analysis method or a dynamic analysis method can be adopted. The static analysis method, which is the one referred to here, is based on codes such as UBC, IBC, AIJ, IS1893 etc. For dynamic analysis, see the sections in this chapter on response spectrum and time history analysis.

Input for the load generation facility consists of two parts:

1. Definition of the load system(s).
2. Generation of primary load cases using previously defined load system(s).

Related Links

- TR.31 Definition of Load Systems (on page 2540)

G.16.1 Moving Load Generator

This feature enables the user to generate static loads on members due to vehicles moving on a structure. Moving load systems consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis. American Association of State Highway and Transportation Officials (AASHTO) vehicles are available within the program and can be specified using standard AASHTO designations.

Related Links

- Add New Vehicle Definitions dialog (on page 3048)
- TR.31.1 Definition of Moving Load System (on page 2541)
- M. To define a vehicle for loading (on page 853)
- Load Generation dialog (on page 3031)
- TR.32.12.1 Generation of Moving Loads (on page 2771)
- M. To generate moving load cases (on page 854)
- TR.31.1 Definition of Moving Load System (on page 2541)
- TR.32.12.1 Generation of Moving Loads (on page 2771)

G.16.2 Seismic Load Generator

The STAAD.Pro seismic load generator follows the procedure of equivalent lateral load analysis explained in UBC, IBC and several other codes. It is assumed that the lateral loads will be exerted in X and Z (or X and Y if Z is up) directions (horizontal) and Y (or Z if Z is up) will be the direction of the gravity loads. Thus, for a building model, Y (or Z if Z is up) axis will be perpendicular to the floors and point upward (all Y (or Z if Z is up) joint coordinates positive). The user is required to set up his model accordingly. Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate equation from the code. IBC 2003, IBC 2000, UBC 1997, 1994, or 1985, IS:1893, Japanese, Colombian and other specifications may be used.
For load generation per the codes, the user is required to provide seismic zone coefficients, importance factors, soil characteristic parameters, etc.

Instead of using the approximate code based formulas to estimate the building period in a certain direction, the program calculates the period using Rayleigh quotient technique. This period is then utilized to calculate seismic coefficient C.

After the base shear is calculated from the appropriate equation, it is distributed among the various levels and roof per UBC specifications. The distributed base shears are subsequently applied as lateral loads on the structure. These loads may then be utilized as normal load cases for analysis and design.

**Related Links**

- [TR.32.12.2.1 Generation of Seismic Loads](on page 2773)
- [TR.32.12.2.2 Generation of IS:1893 Seismic Load](on page 2778)
- [M. To add a seismic load](on page 842)
- [TR.31.2.5 Chinese Static Seismic per GB50011-2001](on page 2563)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.12 IBC 2006/2009 Seismic Load Definition](on page 2592)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.19 Turkish Seismic Code](on page 2613)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.32.4 Area, One-way, and Floor Load Specifications](on page 2664)
- [TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1](on page 2552)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.2 Canadian Seismic Code (NRC) - 1995](on page 2548)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.18 NTC (Normas Técnicas Complementarias) Seismic Load](on page 2610)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.1 RPA (Algerian) Seismic Load](on page 2546)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.17 CFE (Comisión Federal De Electricidad) Seismic Load](on page 2607)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.11 IBC 2000/2003 Load Definition](on page 2588)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.16 Japanese Seismic Load](on page 2605)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.6 Colombian NSR-98 Seismic Load](on page 2571)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2 Definitions for Static Force Procedures for Seismic Analysis](on page 2544)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.21 UBC 1997 Load Definition](on page 2620)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.32.12.2 Generation of Seismic Loads](on page 2773)
- [TR.31.2.20 UBC 1994 or 1985 Load Definition](on page 2617)
G.16.3 Wind Load Generator

The Wind Load Generator is a utility which takes as input wind pressure and height ranges over which these pressures act and generates nodal point and member loads.

This facility is available for two types of structures.

- Panel type or Closed structures
- Open structures

Closed structures are ones like office buildings where non-structural entities like a glass facade, aluminum sheets, timber panels or non-load bearing walls act as an obstruction to the wind. If these entities are not included in the structural model, the load generated as a result of wind blowing against them needs to be computed. So, the steps involved in load generation for such structures are:

1. Identify the panels – regions circumscribed by members so that a polygonal closed area is formed. The area may also be formed between the ground level along one edge and members along the other.
2. Calculate the panel area and multiply it by the wind pressure.
3. Convert the resulting force into nodal point loads.

Plates and solids are not considered in the calculation of the panel area. Openings within the panels may be modeled with the help of exposure factors. An exposure factor is associated with each joint of the panel and is a fractional number by which the area affecting a joint of the panel can be reduced or increased.

The automated load generator should only be used for vertical panels. Panels not parallel to the global Y axis (for Y UP) should be loaded separately.

Open structures are those like transmission towers, in which the region between members is “open” allowing the wind to blow through. The procedure for load generation for open structures is:

1. Calculate the exposed area of the individual members of the model.
2. Multiply that exposed area by the wind pressure to arrive at the force and apply the force on individual members as a uniformly distributed load. It is assumed that all members of the structure within the specified ranges are subjected to the pressure and hence, they will all receive the load. The concept of members on the windward side shielding the members in the inside regions of the structure does not exist for open structures. Members loaded as an open structure need not be vertical.

Related Links
- Create Wind Type Definition dialog (on page 3038)
- TR.31.3 Definition of Wind Load (on page 2623)
- Add New Wind Definitions (data) dialog (on page 3038)
- M. To add a wind load definition (on page 835)
- TR.31.3 Definition of Wind Load (on page 2623)
- TR.32.12.3 Generation of Wind Loads (on page 2779)

G.16.4 Snow Load

STAAD.Pro is capable of generating snow loading on a structure in accordance with the provisions of the ASCE-7-02 code. The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

Related Links
- Snow Load tab (on page 3021)
G.17 Analysis Facilities

Salient features of each type of analysis are discussed in the following sections. Detailed theoretical treatments of these features are available in standard structural engineering textbooks.

Related Links

- **TR.37.1 Linear Elastic Analysis** (on page 2796)

G.17.1 Stiffness Analysis

The stiffness analysis implemented in STAAD.Pro is based on the matrix displacement method. In the matrix analysis of structures by the displacement method, the structure is first idealized into an assembly of discrete structural components.

Structural systems such as slabs, plates, spread footings, etc., which transmit loads in two directions (frame members or finite elements). Each component has an assumed form of displacement in a manner which satisfies the force equilibrium and displacement compatibility at the joints. have to be discretized into a number of three or four noded finite elements connected to each other at their nodes. Loads may be applied in the form of distributed loads on the element surfaces or as concentrated loads at the joints. The plane stress effects as well as the plate bending effects are taken into consideration in the analysis.

Assumptions of the Analysis

For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions:

1. The structure is idealized into an assembly of beam, plate and solid type elements joined together at their vertices (nodes). The assemblage is loaded and reacted by concentrated loads acting at the nodes. These loads may be both forces and moments which may act in any specified direction.

2. A beam member is a longitudinal structural member having a constant, doubly symmetric or near-doubly symmetric cross section along its length. Beam members always carry axial forces. They may also be subjected to shear and bending in two arbitrary perpendicular planes, and they may also be subjected to torsion. From this point these beam members are referred to as “members” in the manual.

3. A plate element is a three or four noded planar element having variable thickness. A solid element is a four-to-eight noded, three dimensional element. These plate and solid elements are referred to as “elements” in the manual.

4. Internal and external loads acting on each node are in equilibrium. If torsional or bending properties are defined for any member, six degrees of freedom are considered at each node (i.e., three translational and three rotational) in the generation of relevant matrices. If the member is defined as truss member (i.e., carrying only axial forces) then only the three degrees (translational) of freedom are considered at each node.

5. Two types of coordinate systems are used in the generation of the required matrices and are referred to as local and global systems.

   Local coordinate axes are assigned to each individual element and are oriented such that computing effort for element stiffness matrices are generalized and minimized. Global coordinate axes are a common datum.
established for all idealized elements so that element forces and displacements may be related to a common frame of reference.

**Basic Equation**

The complete stiffness matrix of the structure is obtained by systematically summing the contributions of the various member and element stiffness. The external loads on the structure are represented as discrete concentrated loads acting only at the nodal points of the structure.

The stiffness matrix relates these loads to the displacements of the nodes by the equation:

\[ A_j = a_j + S_j \cdot D_j \]

This formulation includes all the joints of the structure, whether they are free to displace or are restrained by supports. Those components of joint displacements that are free to move are called degrees of freedom. The total number of degrees of freedom represent the number of unknowns in the analysis.

**Method to Solve for Displacements**

There are many methods to solve the unknowns from a series of simultaneous equations.

In STAAD.Pro, the element stiffness matrices are assembled into a global stiffness matrix by standard matrix techniques used in FEA programs. The technique used by STAAD.Pro was developed based on routines made available in the public domain. The global stiffness matrix is then decomposed as

\[ [K] = [LT] \cdot [D] \cdot [L] \]

which is a modified Gauss method.

\[ [K] \cdot \{d\} = \{F\} \]

becomes

\[ [LT] \cdot [D] \cdot [L] \cdot \{d\} = \{F\} \]

which can be manipulated into a forward and a backward substitution step to obtain \{d\}. STAAD.Pro can detect singular matrices and solve them via a technique copied from Stardyne.

**Basic Solver** An approach which is particularly suited for structural analysis is called the method of decomposition. This method has been selected for use in STAAD. Since the stiffness matrices of all linearly elastic structures are always symmetric, an especially efficient form of the decomposition called Modified Cholesky's method may be applied to these problems. This method is reasonably accurate and well suited for the Gaussian elimination process in solving the simultaneous equations.

**Advanced Solver** (Available effective 2007 Build 01): An approach is used that is mathematically equivalent to the modified Cholesky method. However the order of operations, memory use, and file use is highly optimized. Run times are often 10 to 100 (even 1000) times faster.

**Consideration of Bandwidth**

For the Basic Solver only. The method of decomposition is particularly efficient when applied to a symmetrically banded matrix. For this type of matrix fewer calculations are required due to the fact that elements outside the band are all equal to zero.

STAAD.Pro takes full advantage of this bandwidth during solution, as it is important to have the least bandwidth to obtain the most efficient solution. For this purpose, STAAD.Pro offers features by which the program can internally rearrange the joint numbers to provide a better bandwidth.
For the Advanced Solver only. Internal storage order is automatically calculated to minimize time and memory.

Multiple Structures & Structural Integrity

The integrity of the structure is a very important requirement that must be satisfied by all models. You must make sure that the model developed represents one or more properly connected structures.

An “integral” structure may be defined as a system in which proper "stiffness connections" exist between the members/elements. The entire model functions as one or more integrated load resisting systems. STAAD.Pro checks structural integrity using a sophisticated algorithm and reports detection of multiple structures within the model. If you did not intend for there to be multiple structures, then you can fix it before any analysis. There are several additional model checking options within the Tools and Geometry menus.

Modeling and Numerical Instability Problems

Instability problems can occur due to two primary reasons.

1. Modeling problem

   There are a variety of modeling problems which can give rise to instability conditions. They can be classified into two groups.

   a. Local instability - A local instability is a condition where the fixity conditions at the end(s) of a member are such as to cause an instability in the member about one or more degrees of freedom. Examples of local instability are:

      i. Member Release: Members released at both ends for any of the following degrees of freedom (FX, FY, FZ and MX) will be subjected to this problem.

      ii. A framed structure with columns and beams where the columns are defined as “TRUSS” members. Such a column has no capacity to transfer shears or moments from the superstructure to the supports.

   b. Global Instability - These are caused when the supports of the structure are such that they cannot offer any resistance to sliding or overturning of the structure in one or more directions. For example, a 2D structure (frame in the XY plane) which is defined as a SPACE FRAME with pinned supports and subjected to a force in the Z direction will topple over about the X-axis. Another example is that of a space frame with all the supports released for FX, FY or FZ.

2. Math precision

   A math precision error is caused when numerical instabilities occur in the matrix inversion process. One of the terms of the equilibrium equation takes the form $1/(1 - A)$, where $A = k_1/(k_1+k_2)$; $k_1$ and $k_2$ being the stiffness coefficients of two adjacent members. When a very “stiff” member is adjacent to a very “flexible” member, viz, when $k_1 >> k_2$, or $k_1+k_2 k_1$, $A = 1$ and hence, $1/(1 - A) = 1/0$. Thus, huge variations in stiffnesses of adjacent members are not permitted. Artificially high E or I values should be reduced when this occurs.

   Math precision errors are also caused when the units of length and force are not defined correctly for member lengths, member properties, constants etc.

   You must also ensure that the model defined represents one single structure only, not two or more separate structures. For example, in an effort to model an expansion joint, you may end up defining separate structures within the same input file. Multiple structures defined in one input file can lead to grossly erroneous results.

Related Links

- TR.37 Analysis Specification (on page 2795)
G.17.2 Second Order Analysis

STAAD.Pro offers the capability to perform second order stability analyses.

G.17.2.1 P-Delta Analysis

Structures subjected to lateral loads often experience secondary forces due to the movement of the point of application of vertical loads. This secondary effect, commonly known as the P-Delta effect, plays an important role in the analysis of the structure.

In textbooks this secondary effect is typically referred to as stress stiffening for members in tension (or softening for compression). The stiffness changes due to P-Delta are known as geometric stiffness, \([K_g]\). There are two types of P-Delta effects for members. P-\(\Delta\) which is due to the displacement of one end of a member relative to the other end (e.g., story drift of column members). A second effect is P-\(\delta\) which is due to the bending of the member.

P-\(\delta\) due to the bending of the member not only affects the local & global stiffness, nodal displacements, and member end forces; it also has an additional effect on the section displacements and section moments. The (axial compressive member force) times (the local relative to the ends section displacement) gives a section moment in addition to the flexural moment. This additional section moment will cause an additional sectional displacement; and so on. Normally this process will converge after 5-20 iterations if the member buckling load is not exceeded. STAAD.Pro uses up to 20 iterations unless convergence or divergence occurs.

P-\(\delta\) due to the bending of the member can also occur with tension if the member has sufficient bending. STAAD only iterates once for tension.

STAAD.Pro does not include the effects of geometric stiffness for solids. If the part of the structure that deforms involves non-trivial motions of solids, then the results will be erroneous for P-Delta analysis (as well as for buckling analysis).

**Related Links**

- [P Delta Analysis tab](#)
- [TR.37.2 P-Delta Analysis Options](#)
- [A. To specify a P-Delta analysis](#)

G.17.2.1.1 P-Delta Analysis – Large Delta and Small Delta

In STAAD.Pro, a procedure has been adopted to incorporate the P-Delta effect into the analysis without re-forming and factorizing the global stiffness matrix on each iteration. Actually, only the global stiffness matrix is formed and factorized; which must be done for any analysis. Only the relatively fast forward and backward substitution step for typically five to 25 iterations must be performed. This step is done simultaneously for however many cases are being solved. See [G.17.2.1.2 P-Delta Kg Analysis](#) for an alternate formulation of P-Delta that may be used in dynamics.

**Note:** This feature is available in STAAD.Pro 2007 Build 01 and greater.

If a structure is heavily loaded it may become unstable for some load cases. It may take 10 to 30 iterations for this instability to become obvious by the maximum displacements or bending moment envelope values becoming very large or infinite or reported as NaN (“not a number”).

The procedure consists of the following steps:

1. First, the primary deflections are calculated based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. By default, the small delta effects are calculated. To include only the large delta effects, enter the LARGEDELTA option on the PDELT command. These forces and stresses are used to calculate geometric stiffness terms. These terms times the displacement results from the prior iteration create the P-Delta secondary loading. This secondary loading is then combined with the originally applied loading to create the effective load vector for the next iteration.

The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility has been created with this requirement in mind. This facility allows you to combine previously defined primary load cases to create a new primary load case.

3. The revised load vector is used with the static triangular factorized matrix to generate new deflections.

4. Element/Member forces and support reactions are calculated based on the new deflections.

Repeat steps 2 to 4 for several iterations. Three to 30 iterations are recommended. This procedure yields reasonably accurate results with small displacement problems. You are allowed to specify the number of iterations. If the Converged option is used, then set the displacement convergence tolerance by entering a SET PDELTATOL command before the Joint Coordinates. If all changes in displacement dof from one iteration to the next is less than the specified tolerance value, the case is converged.

The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta has the most effect in structures where there are vertical and horizontal loads in the same load case.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits large buckling displacements if the loads make the structure unstable. You may need to repeat the analysis with only one to five iterations in order to get a pre-collapse solution in order to view the large displacement areas.

The section moment due to tension and the section displacements due to shear/bending are added to the moment diagram, if small delta is selected. This is no iteration performed for this step.

Related Links

• TR.37.2 P-Delta Analysis Options (on page 2797)

G.17.2.1.2 P-Delta Kg Analysis

In STAAD.Pro, an alternate procedure has been adopted to incorporate the P-Delta effect into the analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg].

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.

2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

This procedure yields reasonably accurate results with small displacement problems. STAAD.Pro allows the user to specify multiple iterations of this P-Delta-KG procedure; however one iteration is almost always sufficient.
The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only one to five iterations or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.

Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case, a message is printed and the results of the case are set to zero. (In this case, repeat the analysis using PDELTA 30 ANALYSIS SMALLDELTAS instead).

Related Links
- TR.37.2 P-Delta Analysis Options (on page 2797)

G.17.2.1.3 P-Delta K+Kg Dynamic Analysis

In STAAD.Pro, an alternate procedure has been adopted to incorporate the P-Delta effect into dynamic analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg].

**Note:** This feature is available in STAAD.Pro 2007 Build 01 and greater.

This method uses the resulting [K+Kg] matrices from the last static case before the PDELTA KG command in the dynamic cases that precede the PDELTA KG command.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading for case n.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

The final triangular factorization for case n is then used in the dynamic case n+1 along with the masses specified in case n+1 to solve the dynamic analysis.

Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only one to five iterations or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.
Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case a message is printed and the results of the case are set to zero. The dynamic results should be ignored if this type of failure should occur.

Related Links
- TR.37.2 P-Delta Analysis Options (on page 2797)

G.17.2.1.4 AISC 360 Direct Analysis

AISC 360-05 Appendix 7 describes a method of analysis, called Direct Analysis, which accounts for the second order effects resulting from deformation in the structure due to applied loading, imperfections and reduced bending stiffness of members due to the presence of axial load.

The ANSI/AISC 360-05 Direct Analysis procedure has been adopted to incorporate the P-Delta effect into a static analysis by combining the global stiffness matrix and the global geometric stiffness matrix $[K+K_g]$; plus flexural stiffness reduction; plus axial stiffness reduction; plus an additional flexure reduction if member axial compression forces are above 50% of yield; plus the addition of notional loads.

Note: This feature is available in STAAD.Pro 2007 Build 03 and greater.

This is a nonlinear, iterative analysis as the stiffness of the members is dependent upon the forces generated by the load.

1. The primary deflections are calculated by linear static analysis based on the provided external loading for case $n$. The stiffness reductions and notional loads are included here.

2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These forces and stresses are used to calculate geometric stiffness terms. These terms times the displacement results from the prior iteration create the P-Delta secondary loading. This secondary loading is then combined with the originally applied loading to create the effective load vector for the next iteration.

3. The final triangular factorization for case $n$ is then used to calculate displacements and member forces.

   Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility has been created with this requirement in mind. This facility allows you to combine previously defined primary load cases to create a new primary load case.

4. The axial force is compared to yield force to calculate $\tau_b$ (see Appendix 7 of AISC 360-05). Flexure stiffness of selected members is set to $(0.80 \times \tau_b \cdot EI)$

5. Steps 2 to 4 are repeated until convergence or the iteration limit is reached.

   The analysis will iterate; in each step, changing the member characteristics until the maximum change in any $\tau_b$ is less than the specified $\tau$ tolerance. If the maximum change in any $\tau_b$ is less than 100 times the $\tau$ tolerance and the maximum change in any displacement degree of freedom is less than the specified displacement tolerance; then the solution has converged for this case.

General Procedure for Using the Direct Analysis Feature

There are three general steps required to set up a Direct Analysis:

1. Specify the definition with the DEFINE DIRECT command.

2. Specify the notional loads using the NOTIONAL LOAD load specification.

3. Specify a direct analysis method with the PERFORM DIRECT ANALYSIS command. Within this command, you will specify the tolerances or iteration limits.
G.17.2.2 Buckling Analysis

In STAAD.Pro, two procedures have been adopted to incorporate the calculation of the Buckling Factor for a load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

**Note:** This feature is available in STAAD.Pro 2007 Build 01 and greater.

STAAD.Pro does not include the effects of geometric stiffness for solids. If the part of the structure that deforms during buckling involves non-trivial motions of solids, then the results will be erroneous for buckling (as well as for P-Delta analysis).

### Related Links

- TR.37.4 Buckling Analysis (on page 2804)
- Perform Buckling Analysis tab (on page 3074)
- A. To specify buckling analysis (on page 932)
- TR.37.4 Buckling Analysis (on page 2804)

#### G.17.2.2.1 Buckling Analysis - Iterative Method

In STAAD.Pro, a simple procedure has been adopted to incorporate the calculation of the Buckling Factor for any number of primary load cases. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are multiplied by the estimated BF (buckling factor) and then added to the global stiffness matrix K.

   Buckling Kg matrix effects are calculated for frame members and plate elements only. They are not calculated for solid elements. So buckling analysis is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.
3. For compressive cases, the $K_g$ matrix is negative definite. If the buckling factor is large enough, then $[K + BF \times K_g]$ will also be negative definite which indicates that BF times the applied loads is greater than the loading necessary to cause buckling.

4. STAAD.Pro starts an iterative procedure with a BF estimate of 1.0. If that BF causes buckling, then a new, lower BF estimate is used in the next trial. If the BF does not cause buckling, then a higher BF estimate is used. On the first iteration, if the determinant of the $K$ matrix is positive and lower than the determinant of the $K+K_g$ matrix, then the loads are in the wrong direction to cause buckling; and STAAD.Pro will stop the buckling calculation for that case.

5. After a few iterations, STAAD.Pro will have the largest BF that did not cause buckling (lower bound) and the lowest BF that did cause buckling (upper bound). Then each trial will use a BF estimate that is halfway between the current upper and lower bounds for BF.

6. After the default iteration limit is reached or the user specified iteration limit, $\text{MAXSTEPS}$, is reached or when two consecutive BF estimates are within 0.1% of each other; then the iteration is terminated.

7. Results for this load case are based on the last lower bound BF calculated.

   - Only primary load cases may be solved
   - Any number of buckling cases may be solved.
   - Only the first buckling mode (lowest BF) is calculated.
   - The buckling shape may not be as expected even though the buckling factor is OK. To enhance the mode shape result, apply small loads in the locations and directions where you expect the large displacements.

**Note:** During the buckling analysis using the iterative method, if the determinant of the matrix changes sign in one or more steps, then the resulting values of displacements and forces for the last successful step may not be accurate. If using STAAD.Pro Advanced, then it is recommended to check the buckling factor using the eigen method.

### G.17.2.2.2 Buckling Analysis - Eigen Method

In STAAD.Pro, a second procedure has been adopted to incorporate the calculation of the Buckling Factor for one primary load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure. This procedure is an eigenvalue calculation to get buckling factors and buckling shapes.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.

2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects for members are calculated. These terms are the terms of the $K_g$ matrix.

3. An eigenvalue problem is formed. $| [K] - BF_i [K_g] | = 0$

There will be up to 4 buckling factors (BF) and associated buckling mode shapes calculated. The buckling factor is the amount by which the static load case needs to be multiplied by to just cause buckling (Euler buckling). BF less than 1.0 means that the load causes buckling; greater than 1.0 means buckling has not occurred. If BF is negative, then the static loads are in the opposite direction of the buckling load.

**Notes**

- Solid elements do not contribute to $K_g$ in STAAD.Pro.
- Buckling shapes for the last buckling case only may be displayed in the postprocessor. If there are several buckling cases, then all will have their buckling factors printed.
- The displacement and member/element results are not calculated for the load case times the buckling factor.
G.17.2.3 Static Geometrically Nonlinear Analysis

In STAAD.Pro, a procedure has been adopted to incorporate the geometric nonlinearities into the analysis by updating the global stiffness matrix and the global geometric stiffness matrix \([K+K_g]\) on every step based on the deformed position. The deformations significantly alter the location or distribution of loads, such that equilibrium equations must be written with respect to the deformed geometry, which is not known in advance.

**Note:** This feature is available in STAAD.Pro 2007 Build 05 and greater.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.
3. Next the deflections are re-calculated. Now equilibrium is computed in the deformed position to get out of balance forces. The tangential stiffness matrix is determined from each members new position; the Kg matrix is updated; and the out of balance forces are applied to get the next iteration result.
4. Repeat until converged. If displacements are much too large, then try using ARC 5 to limit displacements on the first linear static step to 5 inches or some suitable value. The STEP 10 parameter may help by loading the structure over many steps.
5. The options for Newton-Raphson, Kg, Steps = 1 are usually taken; but these options are available for some difficult cases.
6. Offset beams, curved beams, cables are not permitted. Tension/compression is not permitted.

Nonlinear effects are calculated for springs, frame members and plate elements only. They are not calculated for solid elements.

The maximum displacement should be reviewed for Nonlinear analyses because this analysis type may result in buckling or large displacements.

The following limitations should be noted regarding static, geometrically nonlinear analyses:

- Large rotations in one step should be avoided by using more steps.
- Very large displacements, unstable structures, and/or post-buckling should be avoided.
- Geometrically nonlinear only. No tension/compression or contact is considered. No yield, plastic moment hinges or bilinear behavior is considered.
- Solids cannot be used for this analysis method.

**Note:** The nonlinear analysis command requires the Advanced Analysis Engine package.

**Related Links**

- Nonlinear Analysis tab (on page 3071)
- TR.37.8 Geometric Nonlinear Analysis (on page 2830)
- A. To specify a nonlinear analysis (on page 930)
- TR.37.8 Geometric Nonlinear Analysis (on page 2830)

G.17.2.4 Imperfection Analysis

Structures subjected to vertical and lateral loads often experience secondary forces due to curvature imperfections in the columns and beams. This secondary effect is similar to the P-Delta effect. In STAAD.Pro the procedure consists of the following steps:
1. First, the deflections and the axial forces in the selected imperfect members are calculated based on the provided external loading.
2. The axial forces and the input imperfections are then used to compute an additional loading on the selected imperfect members that are in compression. These additional loads are combined with the originally applied loading.
3. The static analysis is performed with the combined loading to obtain the final result.

The section moment due to tension and the section displacements due to shear/bending are added to the moment diagram, if small delta is selected. This is no iteration performed for this step.

Related Links
- TR.37.9 Imperfection Analysis (on page 2832)
- Perform Imperfection Analysis tab (on page 3073)
- A. To specify an imperfection analysis (on page 932)
- TR.26.6 Member Imperfection Information (on page 2511)
- TR.37.9 Imperfection Analysis (on page 2832)

G.17.2.5 Multilinear Analysis

When soil is to be modeled as spring supports, the varying resistance it offers to external loads can be modeled using a multilinear analysis.

An example is when the soil’s behavior in tension differs from its behavior in compression. Stiffness-Displacement characteristics of soil can be represented by a multi-linear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. The load cases in a multi-linear spring analysis must be separated by the CHANGE command and PERFORM ANALYSIS command. The SET NL command must be provided to specify the total number of primary load cases. There may not be any PDELTA, dynamic, or TENSION/COMPRESSION member cases. The multi-linear spring command will initiate an iterative analysis which continues to convergence.

Related Links
- TR.27.4 Multilinear Spring Support Specification (on page 2520)

G.17.2.6 Tension- and Compression-Only Analysis

When some members or support springs are linear but carry only tension or only compression, then this analysis may be used.

This analysis is automatically selected if any member or spring has been given the tension or compression only characteristic. This analysis is an iterative analysis which continues to convergence. Any member/spring that fails its criteria will be inactive (omitted) on the next iteration. Iteration continues until all such members have the proper load direction or are inactive (default iteration limit is 10).

This is a simple method that may not work in some cases because members are removed on interim iterations that are needed for stability. If instability messages appear on the second and subsequent iterations that did not appear on the first cycle, then do not use the solution. If this occurs on cases where only springs are the tension/compression entities, then use multilinear spring analysis.

Note:

If the list of members that are to be considered as tension changes between load cases (which would be characterized with the inclusion of an additional MEMBER TENSION command with the revised member list),
then this should be followed by an additional analysis command (and a CHANGE command) before starting the next load case definition.

If the list of members does not change (i.e., there is no new MEMBER TENSION command defined between these load cases), then it is not necessary to include any additional analysis and change commands, the analysis will automatically reset the axial status and iterate to solve for all the specified tension members.

G.17.2.7 Nonlinear Cable or Truss Analysis

**Note:** This feature is available in limited form.

When all of the members, elements and support springs are linear except for cable and/or preloaded truss members, then this analysis type may be used. This analysis is based on applying the load in steps with equilibrium iterations to convergence at each step. The step sizes start small and gradually increase (15-20 steps is the default). Iteration continues at each step until the change in deformations is small before proceeding to the next step. If not converged, then the solution is stopped. You can then select more steps or modify the structure and rerun.

Structures can be artificially stabilized during the first few load steps in case the structure is initially unstable (in the linear, small displacement, static theory sense).

The user has control of the number of steps, the maximum number of iterations per step, the convergence tolerance, the artificial stabilizing stiffness, and the minimum amount of stiffness remaining after a cable sags.

This method assumes small displacement theory for all members/trusses/elements other than cables & preloaded trusses. The cables and preloaded trusses can have large displacement and moderate/large strain. Cables and preloaded trusses may carry tension and compression but cables have a reduced E modulus if not fully taut. Pretension is the force necessary to stretch the cable/truss from its unstressed length to enable it to fit between the two end joints. Alternatively, you may enter the unstressed length for cables.

The current nonlinear cable analysis procedure can result in compressive forces in the final cable results. The procedure was developed for structures, loadings, and pretensioning loads that will result in sufficient tension in every cable for all loading conditions. The possibility of compression was considered acceptable in the initial implementation because most design codes strongly recommend cables to be in tension to avoid the undesirable dynamic effects of a slack cable such as galloping, singing, or pounding. The engineer must specify initial pretensioning tensions which will ensure that all cable results are in tension. In addition this procedure is much more reliable and efficient than general nonlinear algorithms. To minimize the compression the SAGMIN input variable can be set to a small value such as 0.01, however that can lead to a failure to converge unless many more steps are specified and a higher equilibrium iteration limit is specified. SAGMIN values below 0.70 generally requires some adjustments of the other input parameters to get convergence.

Currently the cable and truss are not automatically loaded by selfweight, but the user should ensure that selfweight is applied in every load case. Do not enter component load cases such as wind only; every case must be realistic. Member loads will be lumped at the ends for cables and trusses. Temperature load may also be applied to the cables and trusses. It is OK to break up the cable/truss into several members and apply forces to the intermediate joints. Y-up is assumed and required.

The member force printed for the cable is Fx and is along the chord line between the displaced positions of the end joints.

The analysis sequence is as follows:

1. Compute the unstressed length of the nonlinear members based on joint coordinates, pretension, and temperature.
2. Member/Element/Cable stiffness is formed. Cable stiffness is from EA/L and the sag formula plus a geometric stiffness based on current tension.

3. Assemble and solve the global matrix with the percentage of the total applied load used for this load step.

4. Perform equilibrium iterations to adjust the change in directions of the forces in the nonlinear cables, so that the structure is in static equilibrium in the deformed position. If force changes are too large or convergence criteria not met within 15 iterations then stop the analysis.

5. Go to step 2 and repeat with a greater percentage of the applied load. The nonlinear members will have an updated orientation with new tension and sag effects.

6. After 100% of the applied load has converged then proceed to compute member forces, reactions, and static check. Note that the static check is not exactly in balance due to the displacements of the applied static equivalent joint loads.

The load cases in a nonlinear cable analysis must be separated by the CHANGE command and PERFORM CABLE ANALYSIS command. The SET NL command must be provided to specify the total number of primary load cases. There may not be any Multi-linear springs, compression only, PDelta, NONLINEAR, or dynamic cases.

Also for cables and preloaded trusses:

1. Do not use Member Offsets.
2. Do not include the end joints in Master/Slave command.
3. Do not connect to inclined support joints.
4. Y direction must be up.
5. Do not impose displacements.
6. Do not use Support springs in the model.
7. Applied loads do not change global directions due to displacements.
8. Do not apply Prestress load, Fixed end load.
9. Do not use Load Combination command to combine cable analysis results. Use a primary case with Repeat Load instead.

Related Links

- TR.37.3 Nonlinear Cable Analysis (on page 2800)
- TR.23.2 Member Cable Specification (on page 2493)
- TR.37.3 Nonlinear Cable Analysis (on page 2800)
- TR.23.1 Member Truss Specification (on page 2492)

G.17.2.8 Advanced Nonlinear Cable Analysis

When all of the members, elements, and support springs are linear except for cable members, then this analysis type may be used.

**Note:** This feature is available in limited form.

You have control of the number of steps, the maximum number of iterations per step, the convergence tolerance, include/exclude Kg matrix and use full/modernized Newton-Raphson method.

The nonlinear static solver employs the Newton method (full Newton or modified Newton method) to analyze nonlinear problems. In STAAD.Pro, cable elements and P-Delta effect in beam/column and plates cause geometric nonlinearity.
Steps Included in the Analysis

The nonlinear static analysis has several steps.

1. Final applied loading vector \( \{ P_{\text{ext}} \} \) is assembled. Incremental load vector \( \{ P \} = \{ P_{\text{ext}} \} / n\text{Steps} \) is calculated where \( n\text{Steps} \) is the number of load steps. \( \{ u_{\text{prev}} \} = \{ 0 \} \) is also defined.

2. For the first iteration, the unbalanced loading \( \{ \Delta P \} = \{ P \} \) and displacement vector \( \{ u \} = \{ u_{\text{prev}} \} \), the stiffness matrix \( [K] \) are assembled. If any elements are performing nonlinearly, their element stiffness matrix will be determined based on their current nonlinear status.

3. The equation \( [K] \{ \Delta u \} = \{ \Delta P \} \) is solved to find out the incremental displacement \( \{ \Delta u \} \).

4. The current displacement vector is calculated as \( \{ u \} = \{ u \} + \{ \Delta u \} \).

5. This newly calculated \( \{ u \} \) is used to update all elements nodal coordinates.

6. Based on the updated elements, the element reaction \( \{ R \} \) is calculated.

7. The unbalanced loading now becomes \( \{ \Delta P \} = \{ P \} - \{ R \} \).

8. Convergence is checked by comparing \( |\{ \Delta P \}| / \{ P \} \) with \( \varepsilon \). If convergence is achieved the current displacement is saved as the displacement of previous iteration, i.e. \( \{ u_{\text{prev}} \} = \{ u \} \). The next load increment is applied and same all steps are repeated.

9. If convergence is not achieved, steps 2-7 are repeated until convergence is achieved or the maximum iteration number is reached.
Theory of Cable Elements

*Figure 246: 3D cable sketch*
The cable element formulation follows the catenary theory. It is a nonlinear element with geometric nonlinearity, but without material nonlinearity. For a 3D cable element shown in Figure 2, the free body equilibrium is:

\[ T \frac{dx}{dp} = -F_1 \]  
\[ T \frac{dy}{dp} = -F_2 \]  
\[ T \frac{dz}{dp} = -F_3 + ws \]  

With geometrical constraint equations and constitutive equations, the relationship between chord length components \( l_x, l_y, l_z \) and end support force components \( F_1, F_2, F_3 \) could be derived as:

\[ l_x = \frac{F_1 L_0}{EA} \cdot \frac{1}{w} \left\{ \ln \left( \sqrt{F_1^2 + F_2^2 + (w L_0 \cdot F_3)^2} \right) + \ln \left( \sqrt{F_1^2 + F_2^2 + F_3^2} \right) \right\} \]  
\[ l_y = \frac{F_2 L_0}{EA} \cdot \frac{1}{w} \left\{ \ln \left( \sqrt{F_1^2 + F_2^2 + (w L_0 \cdot F_3)^2} \right) + \ln \left( \sqrt{F_1^2 + F_2^2 + F_3^2} \right) \right\} \]  
\[ l_z = \frac{F_3 L_0}{EA} + \frac{w L_0}{2EA} + \frac{1}{w} \left\{ \ln \left( \sqrt{F_1^2 + F_2^2 + (w L_0 \cdot F_3)^2} \right) + \ln \left( \sqrt{F_1^2 + F_2^2 + F_3^2} \right) \right\} \]  

Then the derivative express of equation (2) is the flexibility matrix, as shown in equation (3). For brevity, the terms in the equation are not listed.

\[ \begin{bmatrix} \frac{dF_1}{dF_1} & \frac{dF_1}{dF_2} & \frac{dF_1}{dF_3} \\ \frac{dF_2}{dF_1} & \frac{dF_2}{dF_2} & \frac{dF_2}{dF_3} \\ \frac{dF_3}{dF_1} & \frac{dF_3}{dF_2} & \frac{dF_3}{dF_3} \end{bmatrix} \begin{bmatrix} dF_1 \\ dF_2 \\ dF_3 \end{bmatrix} = \begin{bmatrix} f_{11} & f_{12} & f_{13} \\ f_{21} & f_{22} & f_{23} \\ f_{31} & f_{32} & f_{33} \end{bmatrix} \begin{bmatrix} dF_1 \\ dF_2 \\ dF_3 \end{bmatrix} \]

The inverse of the flexibility matrix is the stiffness matrix. So the deformation-force relationship and the stiffness matrix are found. Hence the nonlinear equations can be solved using the finite element analysis solvers.

In the step list of the nonlinear static solver, step 4 will provide the cable element the new updated displacement. And step 2 and step 5 will ask the cable element’s stiffness matrix and reaction forces, which can be calculated with equation (3).

**Limitation of cable elements**

One limitation of using catenary theory is that the cable element cannot be loaded with non-uniformly distributed load, point load inside the element. In order to do such doing, the physical cable must be modeled with multiple analytical cable elements, so that the non-uniform load could be approximately simulated by uniform loads on each analytical cable elements, and there are nodes at locations where point loads are applied.

**Notes**

The load cases in a nonlinear cable analysis must be separated by the CHANGE command and PERFORM CABLE ANALYSIS ADVANCED command. The SET NL command must be provided to specify the total number of primary load cases. There may not be any Multi-linear springs, compression only, PDelta, NONLINEAR, or dynamic cases.

Also for cables:
1. Do not use Member Offsets.
2. Do not include the end joints in Master/Slave command.
3. Do not connect to inclined support joints.
4. Y direction must be up.
5. Do not impose displacements.
6. Do not use Support springs in the model.
7. Applied loads do not change global directions due to displacements.
8. Do not apply Prestress load, Fixed end load.
9. Do not use Load Combination command to combine cable analysis results. Use a primary case with Repeat Load instead.

**Note:** This analysis feature can be used only when Advanced Analysis License is active.

**Related Links**
- TR.23.2 Member Cable Specification (on page 2493)
- TR.37.3 Nonlinear Cable Analysis (on page 2800)

### G.17.3 Dynamic Analysis

Available dynamic analysis facilities include solution of the free vibration problem (eigenproblem), response spectrum analysis, and forced vibration analysis.

**Related Links**
- TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767)
- TR.31.4 Definition of Time History Load (on page 2630)
- TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2721)
- TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2538)
- TR.34 Frequency Calculation (on page 2789)
- TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2538)
- TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2715)
- TR.34 Frequency Calculation (on page 2789)
- TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2538)
- TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994 (on page 2710)
- TR.34 Frequency Calculation (on page 2789)
- TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2539)
- TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688)
- TR.34.2 Modal Calculation Command (on page 2791)
- TR.34.1 Rayleigh Frequency Calculation (on page 2790)
- G.15.1 Joint Loads (on page 2337)
- TR.32.10 Dynamic Loading Specification (on page 2686)
- TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2539)
- TR.30.2 Mode Selection (on page 2540)
- TR.37.6 Steady State and Harmonic Analysis (on page 2807)
- TR.34 Frequency Calculation (on page 2789)
- TR.34.2 Modal Calculation Command (on page 2791)
G.17.3.1 Solution of the Eigenproblem

The eigenproblem is solved for structure frequencies and mode shapes considering a diagonal, lumped mass matrix, with masses possible at all active degrees of freedom (d.o.f) included. Two solution methods may be used: the subspace iteration method for all problem sizes (default for all problem sizes), and the Arnoldi/Lanczos method for evaluating eigenvectors (Advanced Analysis only). Additionally, load dependant Ritz vectors (LDR) can be used for dynamically loaded structures.

For large scale eigen value problems, the Arnoldi method is very efficient.

Autoshifting of Eigenvectors

For large models having a large number of d.o.f and a large number of modes extracted (i.e., memory-bound), an incremental solver mode called autoshifting can be used by the Advanced Analysis. A mode shift value is specified to indicate the fixed number of modes the solution tries to find in each shift. The main benefit of using the incremental solver is that it is memory efficient (e.g., problems that were not solvable before on 32-bit systems might be solved with that technique). It is recommended to be used only when memory allocation failure takes place during eigen solution using subspace-iteration or Arnoldi/Lanczos methods. When the program fails to extract eigenvectors due to insufficient memory, you can use auto-shifting which significantly reduces memory demand.

Note: Generally, the Arnoldi/Lanczos is more robust than subspace iteration with autoshifting. The Arnoldi/Lanczos method is very efficient at finding large eigen vectors with autoshift. Thus, if memory problems occur with subspace-iteration (regular mode), it is recommended to switch to the Arnoldi/Lanczos method (regular mode). If memory problems occur with this method in regular mode, then autoshift can be applied to reduce the memory demand.

Since autoshifting can provide significant reduction in memory demand in the solution, it can be ideal for the eigenvalue solution of large systems. In this case, the you are required to provide a "targeted" number of eigen modes to be searched in each shift. The solution starts with "0" shift and tries to find targeted number of eigen modes. Once completed, a new shift is applied and it tries to find next set of eigenvalues within the new shift. This continues until all required number of eigenvalues are found.

The subspace iteration method is very sensitive to calculated shift resulting in a partial extraction of mode. It is recommended to use this solution as a supplemental or alternative solution to existing (default mode). Whenever the program extracts partial set of eigen vectors, it issues a warning message.

If the solution return partial results, it means that not all the required number of eigen modes were found. But the solution guarantees that no eigen values are missed among returned results. Partial results can be still useable for dynamic analysis if they satisfy other analysis requirements. For example, “m” modes satisfy 90% mass participation.

Partial results can be also returned by the subspace method (Advanced Analysis) without using auto shifting method.

The program may miss some Eigen values because of applied shift while performing Eigen solution using Subspace-iteration method. In this case, a warning message is given. In this case, the results returned include missing modes. These results should be used with caution and it is strongly advised for further investigation (dynamic contribution from missing modes might be too important to ignore in analysis).

When the Arnologi/Lanczos method is used with autoshifting, an initial frequency shift may also be specified.
Load-Dependent Ritz Vectors

Research has indicated that considering the effect of natural free-vibration mode shapes may not be the most efficient basis for mode-superposition analysis of structures subjected to dynamic loads. This implies that dynamic analyses—like response spectrum and time history analysis—based on a special set of load-dependent Ritz vectors may yield more accurate results than the use of the same number of natural modes.

There are several reasons to consider Ritz vector analysis as a more efficient approach.

a. For large structural systems, the solution to find free-vibration modes and frequencies may require a significant amount of computational effort.

b. The Ritz vectors method takes into account the spatial distribution of the dynamic loading, whereas the direct use of natural modes neglects this information. Therefore, many of the natural mode shapes that are calculated may not have significant contribution to the dynamic response.

c. Ritz vector analysis by default does not include static correction due to higher mode truncation. The command M.E is required to be issued to include missing mass correction.

The Ritz vectors method is recommended where the solution with eigen vectors fails to capture 90% mass participation (a mandatory requirement of most country seismic codes) with a reasonable number of modes.

It is also recommended where eigen vectors capture irrelevant modes. Even though they are real modes, they are not relevant to the structural response due to the applied dynamic loading.

The spatial distribution of the dynamic load vector serves as a starting load vector to initiate the analysis process. The program will automatically generate this starting load or you can specify it using the DEFINE STARTING LOAD command. In the program generated method, the starting load vector is generated using the mass model of the structure. Assuming a mass matrix only has translational components, the resulting load vector will have force components in the directions of all the translational degrees of freedom, and will have zero values for all the rotational degrees of freedom. This will guarantee a static deflection with components in many directions with this initial mode assumption. As long as the initial mode assumption has a component in the direction of interest, it will yield a correct set of Ritz vectors.

If you define a set of starting load vectors, then this load vector will have a force component in one translational degree of freedom in the direction of the dynamic load. This results in a static deflection mainly in the direction of interest. In some models where mass participation is predominant in one translational direction for the initial few modes, this method can achieve 90% mass participation with only a few modes.

Note: A single set of Ritz vectors can be derived from a single starting load. If all starting loads that participate in dynamic loads are defined, a single set of Ritz vectors can be extracted from all participating starting loads. Therefore, use of the DEFINE STARTING LOAD command is best suited if the structural response is predominant in one translational direction. If response in multiple translational d.o.f. are predominant, then it is recommended to use the program generated starting load vector.


Related Links
- Add New Define Starting Mass Load dialog (on page 3000)
- TR.31.9 Defining Starting Load (on page 2648)
- M. To use starting vectors with load-dependant Ritz vectors (on page 865)
- TR.5 Set Command Specification (on page 2413)
- TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2539)
- TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2538)
G.17.3.2 Mass Modeling

The natural frequencies and mode shapes of a structure are the primary parameters that affect the response of a structure under dynamic loading. The free vibration problem is solved to extract these values. Since no external forcing function is involved, the natural frequencies and mode shapes are direct functions of the stiffness and mass distribution in the structure. Results of the frequency and mode shape calculations may vary significantly depending upon the mass modeling. This variation, in turn, affects the response spectrum and forced vibration analysis results. Thus, extreme caution should be exercised in mass modeling in a dynamic analysis problem.

In STAAD.Pro, all masses that are capable of moving should be modeled as loads applied in all possible directions of movement. Even if the loading is known to be only in one direction there is usually mass motion in other directions at some or all joints and these mass directions (applied as loads, in weight units) must be entered to be correct. Joint moments that are entered will be considered to be weight moment of inertias (force-length² units).

**Note:** Take care to enter selfweight, joint, and element loadings in global directions with the same sign as much as possible so that the representative masses do not cancel each other.

Member/Element loadings may also be used to generate joint translational masses. Note that loads (representing the masses) defined as member concentrated loads, or partially distributed member loads, on non-globally aligned members may result in additional mass being included in orthogonal directions at the nodes. This is because the resolution of these loads (masses) onto the nodes is only considered in the positive direction, and thus does not account for any directional sign of the effect at the node. Member end joint moments that are generated by the member loading (including concentrated moments) are discarded as irrelevant to dynamics. Enter mass moments of inertia, if needed, at the joints as joint moments.

STAAD.Pro uses a diagonal mass matrix of six lumped mass equations per joint. The selfweight or uniformly loaded member is lumped 50% to each end joint without rotational mass moments of inertia. The other element types are integrated but—roughly speaking—the weight is distributed equally amongst the joints of the element.

The members/elements of finite element theory are simple mathematical representations of deformation meant to apply over a small region. The finite element analysis (FEA) procedures will converge if you subdivide the elements and rerun; then subdivide the elements that have significantly changed results and rerun; and so on, until the key results are converged to the accuracy needed.

An example of a simple beam problem that needs to subdivide physical members to better represent the mass distribution (as well as the dynamic response and the force distribution response along members) is a simple floor beam between two columns will put all of the mass on the column joints. In this example, a vertical ground motion will not bend the beam even if there is a concentrated force (mass) at mid span.

Masses that are assigned to slave degrees of freedom (dof) are moved to the master node with a rotatory mass moment of inertia applied at the master. This will be an approximation if the master node is not at the center of gravity (CG, i.e., center of mass) of the slave masses.

In addition, the dynamic results will not reflect the location of a mass within a member (i.e., the masses are lumped at the joints). This means that the motion, of a large mass in the middle of a member relative to the ends of the member, is not considered. This may affect the frequencies and mode shapes. If this is important to the solution, split the member into two. Another effect of moving the masses to the joints is that the resulting shear/moment distribution is based as if the masses were not within the member.
Note: If one end of a member is a support, then half of the member mass is lumped at the support and will not move during the dynamic response. Use ENFORCED supports to minimize this limitation.

Related Links

- TR.32 Loading Specifications (on page 2650)
- TR.31.6 Defining Reference Load Types (on page 2642)
- TR.28.2 Floor Diaphragm (on page 2526)
- TR.31.6 Defining Reference Load Types (on page 2642)
- TR.31.8.3 Mass Model Using Reference Load (on page 2646)
- TR.31.6 Defining Reference Load Types (on page 2642)
- TR.31.4 Definition of Time History Load (on page 2630)
- TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767)

G.17.3.3 Damping Modeling

Damping may be specified by entering values for each mode (either explicitly or calculated), by using a formula based on the first two frequencies, or by using composite modal damping. Composite modal damping permits computing the damping of a mode from the different damping ratios for different materials (steel, concrete, soil). Modes that deform mostly the steel would have steel damping ratio, whereas modes that mostly deform the soil, would have the soil damping ratio.

Modeling Methods

<table>
<thead>
<tr>
<th>Damping Method</th>
<th>Related STAAD.Pro Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>A single specified value, used by all modes</td>
<td>DAMP</td>
</tr>
<tr>
<td>Composite damping based on values specified for each material which can include the effect of spring damping, if defined.</td>
<td>CDAMP</td>
</tr>
<tr>
<td>Modal damping which is explicitly defined for each mode.</td>
<td>MDAMP</td>
</tr>
<tr>
<td>Modal damping which is calculated for all modes.</td>
<td>MDAMP, using either the CALCULATE or EVALUATE method</td>
</tr>
</tbody>
</table>

Related Links

- TR.37.6.4 Steady Ground Motion Loading (on page 2809)
- TR.26.1 Define Material (on page 2501)
- TR.37.6.4 Steady Ground Motion Loading (on page 2809)
- TR.26.4 Modal Damping Information (on page 2510)
- TR.37.6.5 Steady Force Loading (on page 2810)
- TR.26.1 Define Material (on page 2501)
- TR.37.6.5 Steady Force Loading (on page 2810)
- TR.26.4 Modal Damping Information (on page 2510)
- TR.37.6.6 Harmonic Ground Motion Loading (on page 2812)
- TR.26.1 Define Material (on page 2501)
G.17.3.3.1 Composite Damping

Composite modal damping is based on a weighted average of strain energies in each material, (or element), for each mode (eigenvector).

The critical composite damping term $D_J$ for mode $J$ is computed by:
\[ D_i = \sum_{i=1}^{n} \frac{(\phi_i)^T b_i \cdot [k] \cdot (\phi_i)^T \cdot [K] \cdot (\phi_j)}{b_i \cdot (\phi_j)^T} \]

where

- \( n \) = total number of degrees of freedom
- \( b_i \) = equivalent percent of critical damping associated with component \( i \)
- \( \phi_j \) = mode shape vector for mode \( j \)
- \( k_i \) = stiffness associated with component \( i \)
- \( K \) = stiffness matrix for the system

**Related Links**
- Material Constant dialog (on page 2977)
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)
- M. To assign a composite damping ratio (on page 874)
- TR.26.1 Define Material (on page 2501)
- TR.26.2 Specifying Constants for Members and Elements (on page 2503)

### G.17.3.3.2 Modal Damping

**Explicit Damping**

With the EXPLICIT option, you must provide unique modal damping values for some or all modes. Each value can be preceded by a repetition factor \((rf\ast\text{damp})\) without spaces.

**Example**

DEFINE DAMPING INFORMATION
EXPLICIT 0.03 7*0.05 0.04 - 0.012
END

In the above example, mode 1 damping is .03, modes 2 to 8 are .05, mode 9 is .04, mode 10 (and higher, if present) are 0.012.

If there are fewer entries than modes, then the last damping entered will apply to the remaining modes. This input may be continued to 10 more input lines with word EXPLICIT only on line 1; end all but last line with a space then a hyphen. There may be additional sets of EXPLICIT lines before the END.

**Calculate Damping**

The formula used to calculate the damping for modes \( i = 1 \) to \( N \) per modal frequency based on mass and/or stiffness proportional damping (for CALCULATE) is:

\[ D(i) = \left( \frac{\alpha}{2\omega_i} \right) + \left( \frac{\omega_i \beta}{2} \right) \]

If the resulting damping is greater than MAX, then MAX will be used (MAX=1 by default). If the resulting damping is less than MIN, then MIN will be used (MIN=1.E-9 by default). This is the same damping as \( D = (\alpha M + \beta K) \).

**Example:**

DEFINE DAMPING INFORMATION
CALC ALPHA 1.13097 BETA 0.0013926
END
To get 4% damping ratio at 4 Hz and 6% damping ratio at 12 Hz

<table>
<thead>
<tr>
<th>Mode</th>
<th>Hz</th>
<th>Rad/sec</th>
<th>Damp Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.0</td>
<td>25.133</td>
<td>0.04</td>
</tr>
<tr>
<td>3</td>
<td>12.0</td>
<td>75.398</td>
<td>0.06</td>
</tr>
</tbody>
</table>

\[ D(i) = \left( \frac{\alpha}{2\omega_i} \right) + \left( \frac{\omega_i \beta}{2} \right) \]

\[ 0.04 = \frac{\alpha}{50.266} + 12.567 \beta \]

\[ 0.06 = \frac{\alpha}{150.796} + 37.699 \beta \]

\[ \alpha = 1.13097 \]

\[ \beta = 0.0013926 \]

However they are determined, the \( \alpha \) and \( \beta \) terms are entered in the CALC data above. For this example calculate the damping ratio at other frequencies to see the variation in damping versus frequency.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Hz</th>
<th>Rad/sec</th>
<th>Damp Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.0</td>
<td>25.133</td>
<td>0.040</td>
</tr>
<tr>
<td>3</td>
<td>12.0</td>
<td>75.398</td>
<td>0.060</td>
</tr>
<tr>
<td>2</td>
<td>12.0664</td>
<td>0.05375</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>50.2655</td>
<td>0.04650</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>120.664</td>
<td>0.09200</td>
<td></td>
</tr>
<tr>
<td>4.5</td>
<td>28.274</td>
<td>0.03969</td>
<td></td>
</tr>
</tbody>
</table>

The damping, due to \( \beta \) times stiffness, increases linearly with frequency; and the damping, due to alpha times mass, decreases parabolicly. The combination of the two is hyperbolic.
Evaluate Damping

The formula used for EVALUATE (to evaluate the damping per modal frequency) is:

Damping for the first 2 modes is set to \( d_{\text{min}} \) from input.

Damping for modes \( i = 3 \) to \( N \) given \( d_{\text{min}} \) and the first two frequencies \( \omega_1 \) and \( \omega_2 \) and the \( i^{\text{th}} \) modal frequency \( \omega_i \),

\[
A_1 = \frac{d_{\text{min}}}{\omega_1 + \omega_2} \\
A_0 = A_1 \cdot \omega_1 \cdot \omega_2
\]
\[ D(i) = \left( \frac{A_0}{\omega_i} \right) + (A_1 \times \omega_i) \]

If the resulting damping is greater than the \( d_{\text{max}} \) value of maximum damping, then \( d_{\text{max}} \) will be used.

Example:

```
DEFINE DAMPING INFORMATION
EVALUATE 0.02 0.12
END
```

for \( d_{\text{min}} = 0.02 \), \( d_{\text{max}} = 0.12 \) and the \( \omega_i \) given below:

<table>
<thead>
<tr>
<th>Mode</th>
<th>( \omega_i )</th>
<th>Damping Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3</td>
<td>0.0200</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>0.0200</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.0228568</td>
</tr>
<tr>
<td>N</td>
<td>100</td>
<td>0.1200 (calculated as 0.28605 then reset to maximum entered)</td>
</tr>
</tbody>
</table>

Related Links
- [Modal Damping dialog](on page 3059)
- [TR.26.4 Modal Damping Information](on page 2510)
- [M. To explicitly define damping values for modes](on page 875)
- [M. To evaluate damping for modes](on page 875)
- [TR.26.4 Modal Damping Information](on page 2510)

G.17.3.4 Response Spectrum

This capability allows you to analyze the structure for seismic loading. For any supplied response spectrum (either acceleration vs. period or displacement vs. period), joint displacements, member forces, and support reactions are calculated for each mode used in the spectrum solution. These individual modal responses are combined using one of the square root of the sum of squares (SRSS), the complete quadratic combination (CQC), the ASCE4-98 (ASCE), the ten percent (TEN), or the absolute (ABS) methods to obtain the resultant responses. Results of the response spectrum analysis may be combined with the results of the static analysis to perform subsequent design. To account for reversibility of seismic activity, load combinations can be created to include either the positive or negative contribution of seismic results.

Calculation of Forces and Moments at Intermediate Sections

For static load cases, if there is no load applied within the span of a member, for any given degree of freedom (FX, FY, FZ, MX, MY, and MZ), the force or moment value at intermediate span locations can be calculated by linearly interpolating between the values for that degree of freedom at the start and end nodes of the member.

But for response spectrum load cases, this approach is applied at the individual mode basis following which the modal values are combined using the combination method specified in the input. The details of the procedure are as follows:

For any given member, we define the terms RAP and RBP as:
RAP = The force/moment value of the d.o.f under consideration for mode P at the start node of the member (End A)
RBP = The force/moment value of the d.o.f under consideration for mode P at the end node of the member (End B)

**Note:** RAP and RBP are quantities with signs because these are at the individual mode level.

Using linear interpolation, calculate the value of that d.o.f at each of 11 equally spaced intermediate sections along the member length.

So, we now define the term RIP as the value of the d.o.f under consideration at section location “I” for mode “P.”

If the spectrum solution is based on “N” modes, the resultant value for that d.o.f at section location “I” is obtained as:

\[ \text{SRSS}(R_{I1}, R_{I2}, R_{I3}, R_{I4}, \ldots, R_{IN}) \]

or

\[ \text{CQC}(R_{I1}, R_{I2}, R_{I3}, R_{I4}, \ldots, R_{IN}) \]

or a similar calculation for the other modal combination methods.

The values calculated in the above fashion can then be obtained in the output file using the `PRINT SECTION FORCES` command and in tabular or graphical form in the post processing mode.

**Complete Quadratic Combination Method**

This method was first described in “A Replacement for the SRSS Method in Seismic Analysis” ([Earthquake Engineering and Structural Dynamics, Vol. 9 p.187-192](#)) by E.L. Wilson, et al in 1981. The method used in STAAD.Pro is adapted from the textbook *Three Dimensional Static and Dynamic Analysis Of Structures* by Edward L. Wilson.

Using the CQC method, the peak force is determined using the following:

\[
F = \sqrt[3]{\sum_n \sum_m f_n \rho_{nm} f_m}
\]

where

- \( f_n \) = the modal force associated with mode \( n \)
- \( \rho_{nm} \) = the cross-modal coefficient
- \( r = \omega_n / \omega_m \leq 1.0 \)
- \( \zeta = \) the damping ratio. If you use a DEFINE DAMPING INFORMATION command to define modal damping, that will be used here for the corresponding modes. Otherwise, the constant damping or composite damping for the entire structure will be used.

Similar equations are used to determine displacements and other resultant responses.

**Related Links**

- [TR.32.10.1.1 Response Spectrum Specification - Custom](#) (on page 2688)
- [Spectrum Parameters dialog](#) (on page 3027)
- [Generated Spectrum dialog](#) (on page 3027)
G.17.3.5 Response Time History

STAAD.Pro is equipped with a facility to perform a response history analysis on a structure subjected to time varying forcing function loads at the joints and/or a ground motion at its base. This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. Refer to G.17.3.2 Mass Modeling (on page 2365) for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes. The equilibrium equations are written as

\[ [m]\{x''\} + [c]\{x'\} + [k]\{x\} = \{P(t)\} \]

**Note:** The double-prime notation (""") designates the second derivative (i.e., acceleration) and a prime notation ("\')" designates the first derivative (i.e., velocity).

Using the transformation

\[ \{x\} = \sum_{i=1}^{p} \phi_i q_i \]

The equation for \(\{P(t)\}\) reduces to “p” separate uncoupled equations of the form

\[ q''_i + 2 \xi_i \omega_i q'_i + \omega_i^2 q_i = R_i(t) \]

where

\[ \xi \quad = \quad \text{the modal damping ratio} \]
\[ \omega \quad = \quad \text{the natural frequency for the } i^{th} \text{ mode.} \]

These are solved by the Wilson-\(\theta\) method which is an unconditionally stable step by step scheme. The time step for the response is entered by you or set to a default value, if not entered. The \(q_i\)s are substituted in equation 2 to obtain the displacements \(\{x\}\) at each time step.

Time History Analysis for a Structure Subjected to a Harmonic Loading

A Harmonic loading is one which can be described using the following equation:

\[ F(t) = F_0 \sin(\omega t + \phi) \]

where

\[ F(t) \quad = \quad \text{Value of the forcing function at any instant of time } \text{“}t\text{”} \]
\[ F_0 \quad = \quad \text{Peak value of the forcing function} \]
\[ \omega \quad = \quad \text{Frequency of the forcing function} \]
\[ \phi \quad = \quad \text{Phase angle} \]
The results are the maximums over the entire time period, including start-up transients. So, they do not match steady-state response.

**Definition of Input in STAAD.Pro for the above Forcing Function**

As can be seen from its definition, a forcing function is a continuous function. However, in STAAD.Pro, a set of discrete time-force pairs is generated from the forcing function and an analysis is performed using these discrete time-forcing pairs. What that means is that based on the number of cycles that you specify for the loading, STAAD.Pro will generate a table consisting of the magnitude of the force at various points of time. The time values are chosen from this time '0' to \( n \times tc \) in steps of “STEP” where \( n \) is the number of cycles and \( tc \) is the duration of one cycle. STEP is a value that you may provide or may choose the default value that is built into the program. STAAD.Pro will adjust STEP so that a 1/4 cycle will be evenly divided into one or more steps. See TR.31.4 Definition of Time History Load (on page 2630) for a list of input parameters that need to be specified for a Time History Analysis on a structure subjected to a Harmonic loading.

The relationship between variables that appear in the STAAD.Pro input and the corresponding terms in the equation shown above is explained below.

\[
F_0 = \text{Amplitude} \\
\omega = \text{Frequency} \\
\phi = \text{Phase}
\]

Forces applied at slave dof will be ignored; apply them at the master instead.

**Related Links**

- TR.31.4 Definition of Time History Load (on page 2630)
- Add New Time History Definitions dialog (on page 3056)
- M. To define a time history type from tabular data (on page 855)
- M. To define a time history type from a function (on page 856)
- M. To define a time history type by spectrum (on page 857)
- M. To define a time history type by external file (on page 859)
- Define (Time History) Parameters dialog (on page 3058)
- TR.31.4 Definition of Time History Load (on page 2630)
- M. To define time history parameters (on page 860)
- Time History tab (on page 3017)
- TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767)
- M. To add a time history load (on page 861)
- TR.31.4 Definition of Time History Load (on page 2630)
- TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767)

**G.17.3.6 Steady State and Harmonic Response**

A structure [subjected only to harmonic loading, all at a given forcing frequency and with non-zero damping] will reach a steady state of vibration that will repeat every forcing cycle. This steady state response can be computed without calculating the transient time history response prior to the steady state condition.

\[
R(t) = R_0 \sin(\omega t + \phi)
\]

The result, \( R \), has a maximum value of \( R_0 \) and a phase angle \( \phi \). These two values for displacement, velocity, and acceleration at each joint may be printed or displayed.
This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. See G.17.3.2 Mass Modeling (on page 2365) for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the “p” lowest modes.

A Harmonic loading is one in which can be described using the following equation:

\[ F(t) = F_0 \sin(\omega t + \phi) \]

where

- \( F(t) \) = Value of the forcing function at any instant of time “t”
- \( F_0 \) = Peak value of the forcing function
- \( \omega \) = Frequency of the forcing function
- \( \phi \) = Phase angle

![Figure 249: Plot of the harmonic loading function](image)

The results are the steady-state response which is the absolute maximum of displacement (and other output quantities) and the corresponding phase angle after the steady state condition has been reached.

In addition, a Harmonic response can be calculated. This response consists of a series of Steady State responses for a list of frequencies. The joint displacement, velocity, or acceleration can be displayed as the response value versus frequency. Load case results are the maximums over all of the frequencies.

All results are positive as in the Response Spectrum and Time history analyses. This means section results should be ignored (BEAM 0.0 in Parameters for code checking). Because of this, you may want to add the steady state response to Dead & Live loads for one combination case and subtract the steady state response from those loads for another combination case.

Ground motion or a joint force distribution may be specified. Each global direction may be at a different phase angle.

Output frequency points are selected automatically for modal frequencies and for a set number of frequencies between modal frequencies. There is an option to change the number of points between frequencies and an option to add frequencies to the list of output frequencies.

The load case that defines the mass distribution must be the case just before the **PERFORM STEADY STATE ANALYSIS** command. Immediately after that command is a set of data starting with **BEGIN STEADY** and ending with **END STEADY**. The list of additional frequencies and the steady state load cases with joint loads or ground accelerations and phasing data are entered here. The optional print command for the maximum displacement and associated phase angle for selected joints must be at the end of this block of input.

**Note:** Stardyne-Dynre2 data beginning with **START2** and ending with **ALL DONE** may substitute for the BEGIN to END **STEADY** data if the STRESS data is omitted.

**Note:** A license for the advanced analysis module is required to access this feature.
G.17.4 Pushover Analysis

Pushover analysis is a static, nonlinear procedure using simplified nonlinear technique to estimate seismic structural deformations. It is an incremental static analysis used to determine the force-displacement relationship, or the capacity curve, for a structure or structural element.

In STAAD.Pro, the basis for this analysis is the information published in the documents FEMA 356 : 2000 and ATC 40.

**Note:** A license for the advanced analysis module is required to access this feature.

G.17.4.1 Overview of Pushover Analysis

This section includes a an overview of the pushover analysis procedure as well as some of the concepts involved with how STAAD.Pro implements this form of analysis.

G.17.4.1.1 What is a Pushover Analysis?

A pushover analysis is a static, nonlinear procedure using a simplified, nonlinear technique to estimate seismic structural deformations. It is an incremental static analysis which is used to determine the force-displacement relationship – or the capacity curve – for a structure or structural element.

The analysis involves applying horizontal loads, in a prescribed pattern, to the structure incrementally, i.e., pushing the structure and plotting the total applied shear force and associated lateral displacement at each increment, until the structure reaches a collapse condition or a prescribed limit.

This definition of a pushover analysis is in accordance with both the following references:


G.17.4.1.2 Purpose of a Pushover Analysis

It is expected that most buildings rehabilitated in accordance with a standard, would perform within the desired levels when subjected to the design earthquakes. Structures designed according to the existing seismic codes provide minimum safety to preserve life and in a major earthquake, they assure at least gravity-load-bearing elements of non-essential facilities will still function and provide some margin of safety. However, compliance with the standard does not guarantee such performance. They typically do not address performance of non-structural components neither provide differences in performance between different structural systems. This is because it cannot accurately estimate the inelastic strength and deformation of each member due to linear elastic analysis. Although an elastic analysis gives a good indication of elastic capacity of structures and indicates
where first yielding will occur, it cannot predict failure mechanisms and account for redistribution of forces during progressive yielding.

To overcome this disadvantages different nonlinear static analysis method is used to estimate the inelastic seismic performance of structures, and as the result, the structural safety can be secured against an earthquake. Inelastic analysis procedures help demonstrate how buildings really work by identifying modes of failure and the potential for progressive collapse. The use of inelastic procedures for design and evaluation is an attempt to help engineers better understand how structures will behave when subjected to major earthquakes, where it is assumed that the elastic capacity of the structure will be exceeded. This resolves some of the uncertainties associated with code and elastic procedures.

The practice of earthquake engineering is rapidly evolving to understand the behavior of buildings subjected to strong earthquakes. In order to be able to predict such behavior pushover analysis is performed. The overall capacity of a structure depends on the strength and deformation capacities of the individual components of the structure. In order to determine capacities beyond the elastic limit some form of nonlinear analysis, like Pushover Analysis, is required. It is a new performance based seismic design to achieve analytically a structural design that will reliably perform in a prescribed manner under one or more seismic environment.

G.17.4.1.3 Objective of a Pushover Analysis

Pushover analysis is a performance-based analysis that refers to a methodology in which structural criteria are expressed in terms of achieving a performance objective. This is contrasted to the conventional method in which structural criteria are defined by limits on member forces resulting from a prescribed level of applied shear force.

A performance level describes a limiting damage condition which may be considered satisfactory for a giving building and a given ground motion. The limiting condition is described by the physical damage within the building, the threat to life safety of the building’s occupants created by the damage, and the post earthquake serviceability of the building. The basic approach is to improve the probable seismic performance of the building or to otherwise reduce the existing risk to an acceptable level.

Two key elements of a performance-based design procedure are demand and capacity. Demand is the representation of earthquake ground motion or shaking that the building is subjected to. In nonlinear static analysis procedures, demand is represented by an estimation of the displacements or deformations that the structure is expected to undergo. Capacity is a representation of the structure’s ability to resist the seismic demand. The performance is dependent on the manner that the capacity is able to handle the demand. In other words, the structure must have the capacity to resist demands of the earthquake such that the performance of the structure is compatible with the objectives of the design. Performance objective is to obtain a desired level of seismic performance of the building, generally described by specifying maximum allowable (or acceptable) structural or nonstructural damage, for a specified level of seismic hazard.

There are two nonlinear procedures using pushover methods:

- **Capacity Spectrum Method**
- **Displacement Coefficient Method**

G.17.4.1.3.1 Capacity Spectrum Method

The objective of the Capacity Spectrum Method is to develop appropriate demand and capacity spectra for the structure and to determine their intersection point. During this process, performance of each structural component is also evaluated.

The demand curve is based on the earthquake response spectrum and the capacity curves are based on static, nonlinear pushover analysis. In pushover analysis, the structure is subjected to increasing levels of load, and the
base shear versus roof displacement of the structure is charted along the way. The capacity spectrum is obtained by transforming the base shear versus roof displacement spectrum into a spectral acceleration versus spectral displacement spectrum. The intersection of an appropriate demand curve with the capacity curve is called the performance point. The performance point defines the estimated base shear and displacement of the structure when subjected to the earthquake represented by the demand curve. The behavior of the structure at the performance point is compared with predefined acceptance criteria to determine if the design objective is met.

Capacity (Pushover) Curve

The structure capacity is represented by a pushover curve, often termed as capacity curve. This represents the lateral displacement as a function of the force applied to the structure. The most convenient way to plot the force displacement curve is by tracking the base shear and roof displacement.

![Figure 250: Roof deflection, $\Delta_{\text{roof}}$ plotted versus base shear, $V$](image)

Capacity Spectrum

It is the representation of spectral acceleration vs. spectral displacement derived from the capacity curve. The base shear and the roof displacement are converted to spectral acceleration and spectral displacement respectively by the procedure defined by FEMA to get the capacity spectrum from capacity curve.

![Figure 251: Spectral displacement, $S_d$ plotted versus spectral acceleration, $S_a$](image)

Conversion of Capacity Curve to Capacity Spectrum

Capacity curve, in terms of base shear and roof displacement, is converted to capacity spectrum, which is a representation of the capacity curve in Acceleration Displacement Response Spectra (ADRS) format (i.e., $S_a$ versus $S_d$).

The required equations for conversion are the following:

$$PF_{1} = \left[ \frac{\sum_{i=1}^{N} \left( w_{i} \phi_{i} \right) / g}{\sum_{i=1}^{N} \left( w_{i} \phi_{i} \right)^2 / g} \right]^{1/2}$$  \hspace{1cm} (1-3-1)
α_1 = \left[ \frac{\sum_{i=1}^{N} \left( w_i \phi_{i1} \right) / g}{\sum_{i=1}^{N} w_i / g \sum_{i=1}^{N} w_i \phi_{i1}^2 / g} \right]^{1/2} \tag{1.3-2}

S_a = VW/\alpha_1 \tag{1.3-3}

S_d = \Delta_{\text{roof}}(PF_{1}\phi_{\text{roof},1}) \tag{1.3-4}

where

- \( PF_1 \) = Modal participation factor for the first natural mode.
- \( \alpha_1 \) = Modal mass coefficient for the first natural mode.
- \( w_i/g \) = Mass assigned to level i.
- \( \phi_{i1} \) = Amplitude of mode 1 at level i.
- \( N \) = Level N, the level which is the uppermost in the main portion of the structure.
- \( V \) = Base shear
- \( W \) = Building dead weight plus likely live loads.
- \( \Delta_{\text{roof}} \) = Roof displacement (V and the associated make up points on the capacity curve).
- \( S_a \) = Spectral acceleration
- \( S_d \) = Spectral displacement (and the associated make up points on the capacity spectrum).

Demand spectrum

This curve is obtained by redrawing the design earthquake response spectra as a curve of spectral acceleration vs spectral displacement.

![Response spectrum](image)

Performance

Once capacity and demand spectra are defined, a performance check can be one. A performance check verifies that structural and nonstructural components are not damaged beyond the acceptable limits of the performance objective for the forces and displacements implied by the displacement demand.

\textit{G.17.4.1.3.2 Displacement Coefficient Method}

The objective of the Displacement Coefficient Method is to find the target displacement which is the maximum displacement that the structure is likely to be experienced during the design earthquake. This is equivalent to the performance point in the Capacity Spectrum method. It provides a numerical process for estimating the displacement demand on the structure, by using a bilinear representation of capacity curve and a series of modification factors, or coefficients, to calculate a target displacement.
The structure, directly incorporating the nonlinear load-deformation characteristics of individual components and elements of the building, is subjected to monotonically increasing lateral loads representing inertia forces in an earthquake until a target displacement is exceeded. The damage state comprises deformations for all elements in the structure. Comparison with acceptability criteria for the desired performance goal leads to the identification of deficiencies for individual elements. Performance check at the expected maximum displacement is done to verify whether the lateral force resistance has not degraded by more than a desired percentage (generally 20%) of the peak resistance and the lateral drifts satisfy limits are per standard code.

Target Displacement, $\delta_t$

The target displacement is calculated as per procedure described in Section 3.3.3.3.2 of FEMA 356: 2000. It is given by the following expression:

$$\delta_t = C_0 C_1 C_2 C_3 S_a \frac{T_e^2}{(4\pi^2)} g$$

where

- $C_0$ = Modification factor to relate spectral displacement to building roof displacement, as determined by Table 3-2 of FEMA 356.
- $C_1$ = Modification factor to relate expected maximum inelastic displacements to displacements calculated for linear elastic response
  - $= 1.5$ for $T_e < 0.1$ sec
  - $= 1.0$ for $T_e \geq T_s$
  - $= \left[ 1.0 + \left( \frac{R - 1}{T_s / T_e} \right) / R \right]$ for $T_e < T_s$

  Value of $C_1$ should not be less than 1.0.

- $T_s$ = Characteristic period of the response spectrum, defined as period associated with transition from constant acceleration segment of the spectrum to the constant velocity segment of the spectrum (to be calculated from demand spectrum)
- $T_e$ = Effective fundamental time period
  - $= T_i (K_i / K_e)^{1/2}$

- $T_i$ = Elastic fundamental period
- $K_i$ = Elastic lateral stiffness of the building
- $K_e$ = Effective lateral stiffness of the building. Taken as equal to the secant stiffness calculated at a base shear force equal to 60% of the effective yield strength of the Structure obtained from bilinear representation of Capacity Curve.
- $R$ = Ratio of elastic strength demand to calculated yield strength coefficient
  - $= \frac{S_y}{V_y W} C_m$

- $V_y$ = Effective yield strength calculated using the capacity curve. For larger elements or entire structural systems composed of many components, the effective yield point represents the point at which a sufficient number of individual components or elements have yielded and the global structure begins to experience inelastic deformation.
- $S_a$ = Response spectrum acceleration, at the effective fundamental period and damping ratio of the building (to be calculated from demand spectrum)
- $W$ = Effective seismic weight
- $C_m$ = Effective mass factor as determined by Table 3-1 of FEMA 356.
- $C_2$ = Modification factor to represent the effect of pinched hysteretic shape, stiffness degradation and strength deterioration on maximum displacement response. Taken from Table 3-3 of FEMA 356 for different
framing systems and structural performance levels. Alternatively, $C_2$ may be taken as 1.0 for a nonlinear procedure.

Modification factor to represent increased displacement due to dynamic P-D effects

\[ C_3 = \begin{cases} 1.0 & \text{for buildings with positive post-yield stiffness} \\ 1.0 + |\alpha| (R - 1)^{3/2} / T_e & \text{for buildings with negative post-yield stiffness} \end{cases} \]

Where \( \alpha \) is the ratio of post-yield stiffness to effective elastic stiffness, where the nonlinear force-displacement relation shall be characterized by a bilinear relation.

Refer to Figure 3-1 of FEMA 356 for idealized force-displacement curves.

G.17.4.1.4 Types of Nonlinearity

Both geometric and material nonlinearities are considered in this static nonlinear pushover analysis.

**Geometric Nonlinearity**

This is a type of nonlinearity where the structure is still elastic, but the effects of large deflections cause the geometry of the structure to change, so that linear elastic theory breaks down. Typical problems that lie in this category are the elastic instability of structures, such as in the Euler bulking of struts and the large deflection analysis of a beam-column member. In general, it can be said that for geometrical non-linearity, an axially applied compressive force in a member decreases its bending stiffness, but an axially applied tensile force increases its bending stiffness. In addition, the P-Delta effect is also included in this concept.

**Material Nonlinearity**

In this type of nonlinearity, the material undergoes plastic deformation. Material nonlinearity can be modeled as discrete hinges at a number of locations along the length of a frame (beam or column) element and a discrete hinge for a brace element as discrete material fibers distributed over the cross-section of the element, or as a series of material points throughout the element.

**Related Links**

- *M. To manually define and assign hinges* (on page 863)
- *TR.37.7.5.1 User-Defined Hinge Property* (on page 2826)
- *TR.37.7.5.2 Assignment of Hinge Property to the Members* (on page 2827)
- *Define Hinge Property tab* (on page 3053)

G.17.4.1.5 Force controlled and deformation controlled actions

Force-controlled refers to components, elements, actions, or systems which are not permitted to exceed their elastic limits. This category of elements, generally referred to as brittle or nonductile, experiences significant degradation after only limited post-yield deformation.

Deformation-controlled refers to components, elements, actions, or systems which can, and are permitted to, exceed their elastic limit in a ductile manner. Force or stress levels for these components are of lesser importance than the amount of deformation beyond the yield point.
G.17.4.1.6 Frame element hinge properties

Discrete hinge properties for frame elements are usually based on FEMA-356 criteria. As per section 5.5.2.2.2 of FEMA 356, in lieu of relationships derived from experiment or analysis, the generalized load deformation curve shown in the figure below, with parameters a, b, c, as defined in tables 1.5.6 and 1.5.7, shall be used for components of steel moment frames. Modification of this curve shall be permitted to account for strain-hardening of components as follows:

a. a strain-hardening slope of 3% of the elastic slope shall be permitted for beams and columns unless a greater strain-hardening slope is justified by test data; and

b. where panel zone yielding occurs, a strain-hardening slope of 6% shall be used for the panel zone unless a greater strain-hardening slope is justified by test data.

- Point A is the origin
- Point B represents yielding. No deformation occurs in the hinge up to point B, regardless of the deformation value specified for point B. The displacement (rotation or axial elongation as the case may be) will be subtracted from the displacements at points C, D and E. Only plastic deformation beyond point B will be exhibited by hinge.
- Point C represents ultimate capacity of plastic hinge. At this point hinge strength degradation begins (hinge starts shedding load) until it reaches point D.
- Point D represents the residual strength of the plastic hinge. Beyond point D the component responds with substantially strength to point E.
• Point E represents total failure. At deformation greater than point E the plastic hinge will drop load to zero.

The parameters $Q$ and $Q_{CE}$ ($Q_y$) in Figure 1-6 are generalized component load and generalized component expected strength, respectively. For beams and columns, $\theta$ is the total elastic and plastic rotation of the beam or column, $\theta_y$ is the rotation at yield. For braces $\Delta$ is total elastic and plastic displacement, and $\Delta_y$ is yield displacement.

Use of equations (1-6-1) and (1-6-2) to calculate the yield rotation, $\theta_y$, where the point of contraflexure is anticipated to occur at the mid-length of the beam or column, respectively, shall be permitted.

For beams:

$$\theta_y = \frac{Z \cdot F_{ye} L_b}{6 \cdot EI_b} \quad (1-6-1)$$

For columns:

$$\theta_y = \frac{Z \cdot F_{ye} L_c}{(6 \cdot EI_c)} (1 - \frac{P}{P_{ye}}) \quad (1-6-2)$$

$Q$ and $Q_{CE}$ are the generalized component load and generalized component expected strength, respectively. For flexural actions of beams and columns, $Q_{CE}$ refers to the plastic moment capacity, which shall be calculated using equations (1-6-3) and (1-6-4):

For beams:

$$Q_{CE} = M_{CE} = Z \cdot F_{ye} \quad (1-6-3)$$

For columns:

$$Q_{CE} = M_{CE} = 1.18 Z \cdot F_{ye} (1 - \frac{P}{P_{ye}}) \quad (1-6-4)$$

where:

- $E$ = Modulus of elasticity
- $F_{ye}$ = Expected yield strength of the material
- $I$ = Moment of inertia
- $L_b$ = Beam length
- $L_c$ = Column length
- $M_{CE}$ = Expected flexural strength of a member or Joint, kip-in.
- $P$ = Axial force of the member
- $P_{ye}$ = Expected axial yield force of the member = $A_g F_{ye}$
- $Q$ = Generalized component load
- $Q_{CE}$ = Generalized component expected strength = Effective expected strength, which is defined as the statistical mean value of yield strengths, $Q_y$, for a population of similar components, and includes consideration of strain hardening and plastic section development.

Related Links

- M. To manually define and assign hinges (on page 863)
- TR.37.7.5.1 User-Defined Hinge Property (on page 2826)
- TR.37.7.5.2 Assignment of Hinge Property to the Members (on page 2827)
- Define Hinge Property tab (on page 3053)
- P. To review pushover beam results (on page 2224)
- Capacity Curve graph (on page 3153)

G.17.4.1.6.1 Calculation of $Q_{CE}$

The expected flexural strength is evaluated by the following means for beam or column elements.

Beams
The strength of elements of structural steel under flexural actions with negligible axial load present is calculated in accordance with this section.

The expected flexural strength, $Q_{CE}$, of beam components is determined using equation 1-6-3 (on page 2382). For nonlinear procedure flexural actions of beams is considered deformation controlled. Permissible plastic rotation deformation is as indicated in equations 1-6-1 (on page 2382) and 1-6-2 (on page 2382) (Ref. Sections 5.5.2.2.2 and 5.5.2.3.2 of FEMA 356).

Columns

This section is used to evaluate flexural and axial strengths of structural steel elements with non-negligible axial load present. These actions shall be considered force controlled.

The lower-bound strength, $Q_{CL}$, of steel columns under axial compression is taken to be the lowest value obtained for the limit states of column buckling, local flange buckling, or local web buckling. The effective design strength or the lower-bound axial compressive strength, $P_{CL}$, shall be calculated in accordance with the LRFD method, taking $\phi = 1.0$ and using the lower-bound strength, $F_{yLB}$ for yield strength. The expected axial strength of a column in tension, $Q_{CE}$ is computed in accordance with equation (1-6-1-1) (Ref. Section 5.5.2.3.2 of FEMA 356):

$$Q_{CE} = T_{CE} = A_c F_{ye}$$  \hspace{1cm} (1-6-1-1)

where

- $A_c$ = Area of column
- $F_{ye}$ = Expected yield strength of column
- $T_{CE}$ = Expected tensile strength of column
- $F_{yLB}$ = Lower-bound yield strength

Flexural loading of columns, with axial loads at a target displacement less than 50% of $P_{CL}$ shall be considered deformation controlled and maximum permissible plastic rotation demands on columns, in radians, shall be as indicated in tables 1.5.1 and 1.5.2, dependent on the axial load present and the compactness of the section.

Flexural loading of columns, with axial loads at the target displacement greater than or equal to 50% of $P_{CL}$, shall be considered force-controlled and shall conform to equation (1-6-1-2) (Ref. Section 5.5.2.4.2 of FEMA 356).

$$P_{UF}(2P_{CL}) + M_x/(m_xM_{CEx}) + M_y/(m_yM_{CEy}) \leq 1.0$$  \hspace{1cm} (1-6-1-2)

where

- $P_{UF}$ = Axial force in the member
- $P_{CL}$ = Lower bound compression strength of the column
- $M_x$ = Bending moment in the member for the x-axis
- $M_y$ = Bending moment in the member for the y-axis
- $M_{CEx}$ = Expected bending strength of the column for the x-axis
- $M_{CEy}$ = Expected bending strength of the column for the y-axis
- $m_x$ = Value of m for the column bending about the x-axis in accordance with Table 5-5 of FEMA 365
- $m_y$ = Value of m for the column bending about the y-axis in accordance with Table 5-5 of FEMA 365

G.17.4.1.7 Elements

Major horizontal or vertical portions of the building’s structural systems that act to resist lateral forces or support vertical gravity loads such as frames, shear walls, frame-walls, diaphragms, and foundations.
Primary Elements
These are structural components or elements that provide a significant portion of the structure’s lateral force resisting stiffness and strength at the performance point. These are the elements that are needed to resist lateral loads after several cycles of inelastic response to the earthquake ground motion.

Secondary Elements
These are structural components or elements that are not, or are not needed to be, primary elements of the lateral load resisting system. However, secondary elements may be needed to support vertical gravity loads and may resist some lateral loads.

G.17.4.1.8 Lateral Load Distribution
Lateral loads can be applied by any one of the following three methods per Section 3.3.3.2.3 of FEMA 356.

Method 1
The vertical distribution of the base shear shall be as specified in this section for all buildings. The lateral load applied at any floor level x shall be determined in accordance with equation (1-8-1) and equation (1-8-2):

\[ F_x = C_{vx} V \]  
\[ C_{vx} = \frac{w_x h x}{\sum_{i=1}^{n} w_i h_i} \]  

where

\[ C_{vx} \] = Vertical distribution factor

\[ k \] = Linear interpolation shall be used to calculate values of for intermediate values of k for intermediate values of T.

= 2.0 for \( T \geq 2.5 \) seconds

= 1.0 for \( T \leq 0.5 \) seconds

Method 2
A vertical distribution proportional to the shape of the fundamental mode in the direction under consideration is performed. Use of this distribution shall be permitted only when more than 75% of the total mass participates in the fundamental mode in the direction under consideration, and the uniform distribution is also used.

\[ F_x = \frac{w_x \phi_x}{\sum_{i=1}^{n} w_i \phi_i} \]  

Method 3
A vertical distribution is performed consisting of lateral forces at each level proportional to the total mass at each level.

\[ F_x = \frac{w_x}{\sum_{i=1}^{n} w_i} \]  

where

\[ C_{ux} \] = Vertical distribution factor

\[ k \] = Linear interpolation shall be used to calculate values of for intermediate values of k for intermediate values of T.

= 2.0 for \( T \geq 2.5 \) seconds

= 1.0 for \( T \leq 0.5 \) seconds
V = User-defined base shear

\( w_i = \) Portion of the total building weight W located on or assigned to floor level i

\( w_x = \) Portion of the total building weight W located on or assigned to floor level x

\( h_i = \) Height (in ft) from the base to floor level i

\( h_x = \) Height (in ft) from the base to floor level x

\( F_x = \) Amplitude of mode a floor level x

Related Links
- M. To add a pushover loading (on page 864)
- TR.37.7.3 Define Loading Pattern (on page 2823)
- TR.37.7.8 Pushover Loading Input (on page 2829)
- Define Loading Pattern tab (on page 3051)

G.17.4.2 Pushover Analysis Engineering Reference

This section contains technical references on pushover analysis and how this has been implemented in STAAD.Pro.

**Note:** An Advanced Analysis license is required to access this feature.

Related Links
- G.17.4 Pushover Analysis (on page 2376)

G.17.4.2.1 Performing Pushover Analysis

Pushover analysis in STAAD.Pro is a static, non-linear procedure in accordance with FEMA 356 specification. Basically, in this method, the magnitude of the lateral push load is increased progressively according to a predefined loading pattern until either loading or the deflection reaches the described level.

Pushover analysis as it is currently implemented in STAAD.Pro is limited in application to buildings that are regular and do not have adverse torsional or multimode effects. The capacity curve is generally constructed to represent the first mode response of the structure based on the assumption that the fundamental mode of vibration is the predominant response of the structure.

**G.17.4.2.1.1 Define Steel Moment and Braced Frames**

Steel moment frames are those frames that develop their seismic resistance through bending of steel beams and columns and moment resisting beam-column connections. Steel braced frames are those frames that develop their seismic resistance primarily through axial forces in the components.

User must specify whether the structure is a moment frame or braced frame. By default the program considers it as moment frame.

**Note:** Currently only fully restraint (FR) moment frame and CBF (Concentric Braced frame) frame are considered.

Related Links
- M. To define general pushover data (on page 862)
- TR.37.7.2.1 Type of Frame (on page 2819)
- Define Input tab (on page 3049)
**G.17.4.2.1.2 Define Gravity Loading**

Gravity loads include dead loads and (typically) most live loads. Dead load can be taken as the calculated structure self-weight without load factors, plus realistic estimates of flooring, ceiling, partition and other structural and nonstructural components. Live loads should be evaluated for each structure; with consideration given to current and expected future occupancies.

**G.17.4.2.1.3 Define Lateral (Push) Loading**

The mathematical model directly incorporating the nonlinear load-deformation characteristics of individual components and elements of the building shall be subjected to monotonically increasing lateral loads representing inertia forces in an earthquake until a target displacement is exceeded.

A static, nonlinear pushover analysis usually requires multiple analyses cases. The first pushover load case is gravity load applied to the structure. The rest of the load cases may apply different lateral loads in terms of push load increments, whatever the case may be.

A pushover case may start from zero initial conditions or it may start from the results at the end of a previous pushover case. Thus, the gravity load case starts from zero initial conditions. The first lateral load case will start from the end of the gravity load case, the second lateral load case will start from the end of the first lateral load case, and so on until the target displacement is exceeded.

**Note:** When defining the incremental push load, you should take care to make its value small enough since larger values of incremental loading can prevent the analysis converging.

The lateral loads shall be applied in both positive and negative directions since it may lead to different results for asymmetric structures.

**Related Links**

- *M. To add a pushover loading* (on page 864)
- *TR.37.7.3 Define Loading Pattern* (on page 2823)
- *TR.37.7.8 Pushover Loading Input* (on page 2829)
- *Define Loading Pattern tab* (on page 3051)

**Lateral Loading Pattern**

Lateral loads should be applied in predetermined patterns that represent predominant distributions of lateral inertial loads during earthquake response.

Distribution of lateral load must be applied to the structure when performing a pushover analysis.

Typically push load is defined in any one of the following:

- **a.** User defined static load pattern
- **b.** User defined base shear to be distributed vertically

Incremental push load $\Delta P$ is calculated by using any of the following two methods:

- **a.** You define Push load. In other words, you specify the incremental push load pattern on the structure by defining lateral load at nodes.

- **b.** Or, you define the base shear which is distributed laterally as per methods described in Section 1.4.1. The lateral load at each floor is again divided by the number of load step increment to get actual push load incremental load. Thus:

  $$\Delta P = \frac{V}{N}$$

  where
\[ V = \text{Lateral load distributed from user defined base shear.} \]
\[ N = \text{Total number of load step} \]

The actual load acting on the structure at any load step \( i = \Delta P_i = \Delta P \cdot S_{pi} \)

where

\[ S_{pi} = \text{Stiffness Parameter at } i^{\text{th}} \text{ iteration} \]

\[ = \text{Slope of the capacity curve at } (i-1)^{\text{th}} \text{ iteration} / \text{Initial slope of the capacity curve} \]

During linear stage (i.e., all members in the structure are linear), the stiffness parameter is 1.0. Whenever any member becomes nonlinear the stiffness parameter decreases since slope of the capacity curve becomes less than that during elastic stage. Thus, actual lateral load acting on the nonlinear structure at any load increment stage is less than that during linear stage.

**Note:** Currently, you must define gravity load case as primary loading. First lateral load case starts from the end of the gravity load case.

### G.17.4.2.1.4 Define Primary/Secondary Elements and Components

Elements and components that provide the capacity of the structure to resist collapse under seismic forces induced by ground motion in any direction shall be classified as primary. Other elements and components shall be classified as secondary.

In a typical building, nearly all elements, including many nonstructural components, will contribute to the building's overall stiffness, mass and damping, and consequently its response to the earthquake ground motion. However, not all these elements are critical to the ability of the structure to resist collapse when subjected to strong ground shaking. The secondary designation will be used when a component or element does not contribute significantly or reliably in resisting earthquake effects because of low lateral stiffness, strength or deformation capacity.

**Note:** Currently all elements are considered as primary elements.

### G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria

At the beginning of the analysis user needs to define properties (Refer Section 1.3) and acceptance criteria for all pushover hinges. The program includes several built-in default hinge properties that are based on FEMA 356 tables 1.6.1 (Table 5-6 Pg 5-40 of FEMA 356) and 1.6.2 (Table 5-7 Pg 5-44 of FEMA 356) for steel structures.

While generating default in-built hinge properties following points have been considered.

- **Columns in axial tension in braced frame with negligible moment** - It is considered as deformation-controlled action as per Table 5-7 Pg 5-44 of FEMA 356. Mz and My moment are negligible.
- **Columns in axial compression in braced frame with negligible moment** - It is considered as force-controlled action as per Table 1-5-1 of FEMA 356. Mz and My moment are negligible.
- **Columns in axial tension in braced and moment frame with moment** - As per FEMA 356 Pg 5-17, for steel columns under axial tension or combined axial tension and bending shall be considered deformation controlled. It will simply act as beam.
- **Columns in axial compression in braced and moment frame with moment** - As per FEMA 356 Pg 5-17, for steel columns under combined axial compression and bending stress, the column shall be considered deformation-controlled for flexural behavior, force-controlled for compressive behavior. i.e.
  a. if \( Fx / Pcl \leq 0.5 \) consider deformation-controlled
b. if \( \frac{F_x}{P_{cl}} > 0.2 \) and \( \frac{F_x}{P_{cl}} \leq 0.5 \) consider deformation-controlled

c. if \( \frac{F_x}{P_{cl}} > 0.5 \) consider force-controlled

where \( F_x \) is axial compressive force and \( P_{cl} \) is lower bound compressive strength of column. (Refer Section 1.6.1 for detail).

- **Beams in axial tension in braced frame with negligible moment** - It is considered as deformation-controlled action as per Table 5-7 Pg 5-44 of FEMA 356. \( M_z \) moment is negligible.

- **Beams in axial compression in braced frame with negligible moment** - It is considered as force-controlled action as per Table 1-5-1 of FEMA 356. \( M_z \) moment is negligible.

- **Beams in axial compression and tension in moment and braced frame with moment** - It is considered as deformation-controlled action as per Table 5-6 Pg 5-40 of FEMA 356.

- **Braces in axial tension and compression in braced frame** - It is considered as deformation-controlled action as per Table 5-7 Pg 5-44 of FEMA 356. For braces in compression out-of-plane is considered.

Please note that while generating built-in hinge properties in STAAD.Pro, the following components are *not* considered:

- Column panel zones
- Fully restrained moment connections
- Partially restrained moment connection
- EBF
- Steel plate shear walls

If any member has releases, the effect in member nonlinear stiffness is considered to be same as that for linear member stiffness.

**Related Links**

- [M. To manually define and assign hinges](#) (on page 863)
- [TR.37.7.5.1 User-Defined Hinge Property](#) (on page 2826)
- [TR.37.7.5.2 Assignment of Hinge Property to the Members](#) (on page 2827)
- [Define Hinge Property tab](#) (on page 3053)

**Define Acceptance Criteria**

The performance of a structure and its components is defined by the acceptance criteria to provide desirable information for evaluation or retrofit. It refers to the specific limiting values for the deformations and loadings, for deformation-controlled and force-controlled components respectively, which constitute for acceptable seismic performance. Following three criteria are there:

| Immediate Occupancy (IO) | The post earthquake structural damage state in which only very limited structural damage has occurred. The basic vertical and lateral force resisting systems of the building retain nearly all of their pre-earthquake characteristics and capacities. The risk of life-threatening injury from the structural failure is negligible |
| Life Safety (LS) | The post earthquake damage state in which significant damage to the structure may have occurred but in which some margin against either total or partial structural collapse remains. |
| Collapse Prevention (CP) | The post earthquake structural damage state in which the building's structural system is on the verge of experiencing partial or total collapse. |

The program considers only moment hinge for beam and column for steel structures. If you do not define hinge properties, the program will consider built-in default hinge properties based on FEMA 356. These automatic hinge properties include both moment and axial hinge.
G.17.4.2.1.6 Define Pushover Analysis Solution Control

Pushover analysis will continue until any of the following three conditions is satisfied:

i. Cumulative base shear is less than or equal to the base shear defined by the user

You must define base shear up till which pushover analysis will be performed since design base shear (specific to particular seismic code) excludes non-linear effect. When the structure is subjected to strong earthquake the actual base shear may be very high compared to the design base shear. Under this condition there is no guarantee that the structure will maintain desired performance level. This option is chosen when the magnitude of base shear is known and the structure will be able to support that load.

ii. Displacement at the control joint in the specified direction exceeds specified displacement

This option is chosen when the amount of displacement is known i.e. how far the structure will move but the amount of base shear that the structure will be subjected to is not known. While defining this option please make sure that the displacement component chosen at the control joint increases monotonically during loading. The control node shall be located at the center of mass at the roof of the building.

iii. The structure becomes unstable

This happens whenever hinge formation is such that it renders the structure on the verge of collapse. If neither base shear nor displacement at control joint is known, define a higher value for both these options. During analysis instability will arise due to collapse of different members and make the structure unstable.

Related Links
- M. To define solution control (on page 865)
- TR.37.7.4 Define Solution Control (on page 2825)
- Define Solution Control tab (on page 3055)

G.17.4.2.1.7 Define Input for Demand Spectrum

Demand Spectrum is generated according to the method described in Section 1.6.1.5 of FEMA 356 : 2000.

The program generates Demand Spectrum for the purpose of finding $T_s$ and $S_a$ (corresponding to $T_e$ for the purpose of calculating target displacement).

Related Links
- M. To define pushover spectral data (on page 863)
- TR.37.7.6 Define Spectral Parameters (on page 2828)
- Define Spectrum Details tab (on page 3053)

Generation of Demand Spectrum

The following criteria is used to develop the demand spectrum for a pushover analysis.

Site Classes

A. Hard rock with average shear wave velocity, $v_s > 5,000$ ft/sec

B. Rock with $2,500$ ft/sec < $v_s < 5,000$ ft/sec

C. Very dense soil and soft rock with $1,200$ ft/sec < $v_s$ ≤ $2,500$ ft/sec or with either standard blow count $N > 50$ or undrained shear strength $s_u > 2,000$ psf

D. Stiff soil with $600$-ft/sec < $v_s$ ≤ $1,200$ ft/sec, with $15 < N \leq 50$, or $1,000$ psf ≤ $s_u < 2,000$ psf

E. Any profile with more than 10 feet of soft clay defined in soil with plasticity index $Pl > 20$, or water content $w > 40$ percent, and $s_u > 500$ psf or a soil profile with $v_s < 600$ ft/sec.

F. Soils requiring site specific evaluations

Adjustment for site class
The design short period spectral response acceleration parameter, \( S_{XS} \), and the design spectral response acceleration parameter at one second, \( S_{X1} \), shall be obtained from equations (2-7-1) and (2-7-2), respectively, as follows:

\[
S_{XS} = F_a S_S \\
S_{X1} = F_v S_1
\]

where 

\[
F_a \text{ and } F_v = \text{site coefficients determined respectively from Tables 1-4 and 10-5 in FEMA 365, based on the site class and the values of the response acceleration parameters } S_S \text{ and } S_1 \text{ for the selected return period.}
\]

General Horizontal Response Spectrum

A general response spectrum as shown in Figure 1-1 of FEMA 365 shall be developed using equations (2-6-3), (2-6-4) and (2-6-5) for spectral response acceleration, \( S_a \), versus structural period, \( T \), in the horizontal direction.

For \( 0 < T < T_s \)

\[
S_a = S_{xs} \left( \frac{5}{B_s} - 2 \right) \frac{T}{T_s} + 0.4
\]

(2-7-3)

for \( T < T_s \)

\[
S_a = S_{XS}/B_S
\]

(2-7-4)

for \( T > T_s \)

\[
S_a = S_{X1} / (B_1 T)
\]

(2-7-5)

where

\[
T_s = \frac{S_{X1} B_S}{S_{XS} B_S}
\]

(2-7-6)

\[
T_0 = 0.2 \frac{T_s}{T_s}
\]

(2-7-7)

Now this response spectrum with \( S_a \) vs \( T \) is converted to demand spectrum with \( S_a \) vs \( S_d \) (in case of Capacity Spectrum method)

**Note:** The program generates Demand Spectrum for the purpose of finding \( T_S \) and \( S_a \) (corresponding to \( T_e \) for the purpose of calculating target displacement).

**G.17.4.2.1.8 Define Any Other Input**

There are several other inputs which may be required for pushover analysis.

See [TR.37.7 Pushover Analysis](on page 2819) for additional details.

**G.17.4.2.1.9 Hinge Formation and Hinge Unloading**

This section describes how the program handles the formation of hinges and the resulting redistribution of load.

**Moment Diagrams With Hinges**

For each load increment, member sectional forces are checked with section capacity in order to check formation of hinge. If sectional force exceeds section capacity, hinge formation starts. This implies member lies on or beyond point B on the load deformation curve. The point B in the load deformation curve denotes the yield point of the hinge. The hinge is assumed to be rigid between points A and B until it yields.

When the hinge reaches the deformation denoted by point C, it begins to lose load carrying capacity. When it reaches the deformation point E, the hinge loses all of its load carrying capacity.
Total numbers of 13 sections along the length of member are scanned for $M_z$ and $M_y$ moments. Maximum moment is located and checked with section capacity. If the sectional force exceeds section capacity the material starts yielding at that particular location and the hinge at this particular position lies on point B on load deformation curve.

![Figure 255: Typical beam moment diagram under gravity load, with super-imposed hinges](image)

If bending moment diagram is like figure 1.8 (this happens at initial stage of load increment when push load is much low as compared to dead load) the chance of forming moment hinge is at two ends and at span of the member (as shown by red dots).

![Figure 256: Typical beam moment diagram under significant lateral load, with super-imposed hinges](image)

If bending moment diagram is like figure 1.9 (this happens when push load is much high as compared to dead load) the chance of forming moment hinge is only at sections at or nearer to two ends of the member (as shown by red dots).

**Method of Hinge Unloading**

When a hinge unloads, the program must find a way to remove the load that the hinge was carrying and redistribute it to the rest of the structure. Hinge unloading occurs whenever force-deformation or moment-rotation curve shows a drop in capacity, from point C to point D. The hinge unloads elastically without any plastic deformation (i.e., parallel to slope A-B).

When the hinge force reaches point C, entire structure is unloaded i.e. the program reverses the load on the whole structure until the hinge is unloaded up to the point D. When the hinge reaches point D, the load is again reversed. Other parts may now pick up the load that was removed from the unloading hinge.

**G.17.4.2.1.10 Performance**

A performance check can be done either by Capacity Spectrum method or Displacement Coefficient Method.

**Note:** The program considers Displacement Coefficient method.

**G.17.4.2.2 Member Stiffness Matrix with Plastic Hinge**

When numbers of springs are connected together in series, the force in each spring is the same. Here we will use the flexibility matrix rather than the stiffness matrix, which will enable us to avoid the intermediate displacements at the spring connections as variable.
Thus write for each spring,

\[ \text{Extension} = \text{flexibility} \times \text{force} \]

and add up all the individual extensions to give

\[ \text{Total extension} = \sum \text{sum of flexibilities} \times \text{force} \]

To demonstrate this idea, we will take one composite member as shown in Fig. This member is assumed to consist of a number of sections whose \( F \) matrices are known.

At first if we consider a case where end \( A \) is fixed and a load \( p_x \) is applied to end \( X \), then by definition the displacement at \( X \) is given \( d_X = e_{AX} = F_{AX} p_X \). This displacement is the sum of the displacements due to the strains in the individual sections.

Let us consider the part \( KX \) of the member, lying between end 2 of section \( k \) and \( X \). The load acting on the end of \( KX \) attached to the section \( k \) is equal to \(-H_{KX}p_X\) and the load acting on end 2 of section \( k \) is \( H_{KX}p_X \). Since \( A \) is fixed and \( AJ \) is considered rigid, end 1 of member \( k \) is effectively fixed, and the displacement of its end 2 is given by \( d_k = e_k = F_k p_{2A} = F_k H_{KX} p_X \). Now \( KX \) is a rigid member, and the displacements of its ends therefore satisfy the equation for rigid-body movement. Thus the displacement of \( X \) due to the strain induced in section \( k \) by the load \( p_X \) at \( X \) is \( d_{Xk} = H_{KX} F_k H_{KX} p_X \). Thus the flexibility matrix \( F_k \) of section \( k \) appears as \( H_{KX} F_k H_{KX} \) when viewed from the point \( X \). Similar analysis may be applied to each section of \( AX \), and adding up all the displacement contributions we obtain the complete flexibility matrix of \( AX \) in the form

\[ F_{AX} = \sum H_{KX} F_k H_{KX} \]

Where the summation extends over all the sections making up the member \( Ax \). Inversion of \( F_{AX} \) gives the matrix \( K_{AX} \) and since the matrix \( H_{AX} \) may be obtained either from the overall equilibrium of the member or by multiplying together all the \( H \) matrices of the sections the other stiffness matrices.

**Semi-Rigid Joint Connections**

When a member is attached to the joints at its ends by flexible connections, each of which transmits a moment proportional to the difference between the rotation of the end of the member and the rotation of the joint to which it is attached. Such joints occur in bolted frames, and in welded frames after the onset of plasticity.
To have a clear understanding, first we will describe it for a two-dimensional plane frame and then we will extend it to three-dimensional space frame.

Let us consider a frame of length $L$ and flexural rigidity $EI$, whose ends 1 and 2 are attached to the rest of the frame by connections, which exert moment’s $4Eik_1/L$ and $4Eik_2/L$ respectively per unit difference in rotation. We ignore the length of each connection.

The displacement vector on the member side of the connection is given by the equation

G.17.4.2.3 Modeling Rules for Pushover Analysis in STAAD.Pro

Pushover analysis takes time. Each nonlinear problem is different. Since it is a step-wise linear analysis, analysis time and results is very much dependent on the incremental push load defined in the input file. Small changes in properties or loading can cause large change in nonlinear response. Hence it is important to consider different loading cases and to perform sensitivity studies on the effect of varying the properties of different structural elements. Analysis results depends on the selection of control node, the selection of lateral load patterns and the determination of fundamental period.

It is better to start with simple analytical model. The analytical model of the building should represent all new and existing components that influence the mass, strength, stiffness, and deformability of the structure at or near performance point. Elements and components shown not to significantly influence the building assessment need not be modeled.

The model should perform as expected under linear static loads and modal analysis. The control node shall be located at the center of mass at the roof of a building. It should be selected in such a way that the displacement component in the direction of lateral load monotonically increases as push load increases. Separate mathematical models representing the framing along two orthogonal axes of the building shall be developed for two-dimensional analysis. A mathematical model representing the framing along two orthogonal axes of the building shall be developed for three-dimensional analysis. Independent analysis must be done while using different methods of lateral load distribution.

Since Pushover analysis depends very much on the incremental push load defined in the input file by using LDSTEP parameter, it is better to give a lower value of LDSTEP parameter (preferably within 25) to start with. If the analysis does not run successfully and indicate warning message to increase LDSTEP parameter, parameter value will have to be increased. However, it may happen that the analysis has ran successfully and achieved the convergence criteria, the program is still issuing warning message to increase the LDSTEP parameter to get more accurate results. The value of LDSTEP parameter can be increased to get more accurate results but note that by how much LDSTEP parameter will be increased and what the exact value it will take depends upon the engineer’s judgement. By looking at the Capacity Curve graph, hinge formation and the hinge status (IO, LS, CP or greater than CP) at different load steps, engineer has to judge whether to increase the LDSTEP parameter or not.
The type of warning message only gives an initial indication to the engineer whether the value of LDSTEP parameter will be increased or not.

*** WARNING : ANALYSIS RESULT IS NOT CONVERGING. UNABLE TO FIND A SOLUTION. TRY TO REDUCE INCREMENTAL PUSH LOAD BY INCREASING LDSTEP PARAMETER.

If the warning message is issued as mentioned above, LDSTEP parameter has to be increased.

*** WARNING : STRUCTURE HAS REACHED AN UNSTABLE EQUILIBRIUM WHERE ALL OF A SUDDEN MEMBERS HAVE STARTED TO FAIL. MORE ACCURATE RESULT MAY BE OBTAINED BY REDUCING INCREMENTAL PUSH LOAD BY INCREASING LDSTEP PARAMETER.

If the warning message is issued as mentioned above, LDSTEP parameter may or may not be increased depending upon engineer's choice.

G.17.4.2.4 Scope of Pushover Analysis in STAAD.Pro

Pushover Analysis in STAAD.Pro will produce valid results only when following conditions are satisfied:

1. Buildings are regular and do not have adverse torsional or multimode effects. The capacity curve is generally constructed to represent the first mode response of the structure based on the assumption that the fundamental mode of vibration is the predominant response of the structure.

2. Fully rigid moment frame and concentric braced frame are considered. If any member release is defined, its effect will be same as that for linear analysis.

3. Only steel structures are considered (i.e., material cannot be concrete).

4. Only straight beam and column members are considered (i.e., no curved members, no plate or surface elements).

5. All members of a structure are considered as primary element. No secondary elements are considered.

6. For built-in FEMA hinge formation, out of six degrees of freedom, hinge formation is considered for bending moment about local z axis of beam. For column, hinge formation is considered for bending moment about local z axis of column or for bending moment about local y axis of column, whichever moment becomes the guiding factor. Depending upon the guiding criteria given in FEMA 356:2000, hinge formation in beam is deformation-controlled and that in column can be deformation-controlled or force-controlled. For braced (i.e. truss) member, axial hinge is considered. Beside moment and axial hinge, no other types of hinge formation (e.g. hinge formation due to shear or hinge formation due to coupled effect of axial force and bi-axial bending moment) are considered.

7. Built-in hinge properties and acceptance criteria for beam and columns given in Tables 5-6 and 5-7 of FEMA 356:2000 for beams, columns, and braces are considered.

8. User defined hinge properties are considered for beam and column for hinge formation for bending moment only. For column, only deformation-controlled action is considered as no axial force is considered for checking force-controlled action in column.

9. Out of five methods for lateral load distribution, three methods have been implemented as per Section 3.3.3.2.3 of FEMA 356:2000. The methods implemented are 1.1, 1.2, and 2.1 of Section 3.3.3.2.3 (See G. 17.4.1.8 Lateral Load Distribution (on page 2385)).

10. Performance point of a structure is calculated based on Displacement Coefficient method to find target displacement.

G.18 Member End Forces

Member end forces and moments in the member result from loads applied to the structure. These forces are in the local member coordinate system. The following figures show the member end actions with their directions.
Figure 259: Member end forces when Global Y is vertical

Figure 260: Member end moments when Global Y is vertical
Figure 261: Member end forces when Global Z is vertical (that is, SET Z_UP is specified)

Figure 262: Member end moments when Global Z is vertical (that is, SET Z_UP is specified)
Stress Zones Due to Bending

Figure 263: Stress zones due to bending about the Y axis (MY) for various section types

Note: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension
Figure 264: Stress zones due to bending about the Z axis (MZ) for various section types

**Note:** Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension.

**Related Links**
- TR.41 Section Specification (on page 2839)
- TR.42 Print Specifications (on page 2840)
- TR.40 Load Envelope (on page 2837)
G.18.1 Secondary Analysis

Solution of the stiffness equations yield displacements and forces at the joints or end points of the member. STAAD.Pro is equipped with the following secondary analysis capabilities to obtain results at intermediate points within a member.

Related Links
- TR.41 Section Specification (on page 2839)
- TR.42 Print Specifications (on page 2840)
- TR.44 Printing Section Displacements for Members (on page 2846)
- TR.45 Printing the Force Envelope (on page 2848)

G.18.1.1 Member Forces at Intermediate Sections

With the SECTION command, you may choose any intermediate section of a member where forces and moments need to be calculated. These forces and moments may also be used in design of the members. The maximum number of sections specified may not exceed five, including one at the start and one at the end of a member. If no intermediate sections are requested, the program will consider the start and end member forces for design. However, of the sections provided, they are the only ones to be considered for design.

Related Links
- TR.41 Section Specification (on page 2839)
- TR.42 Print Specifications (on page 2840)
- TR.40 Load Envelope (on page 2837)

G.18.1.2 Member Displacements at Intermediate Sections

Like forces, displacements of intermediate sections of members can be printed or plotted. This command may not be used for truss or cable members.

Related Links
- TR.41 Section Specification (on page 2839)
- TR.42 Print Specifications (on page 2840)
- TR.44 Printing Section Displacements for Members (on page 2846)
- TR.45 Printing the Force Envelope (on page 2848)

G.18.1.3 Member Stresses at Specified Sections

Member stresses can be printed at specified intermediate sections as well as at the start and end joints. These stresses include:

a. Axial stress, which is calculated by dividing the axial force by the cross sectional area,
b. Bending-y stress, which is calculated by dividing the moment in local-y direction by the section modulus in the same direction,
c. Bending-z stress, which is the same as above except in local-z direction,
d. Shear stresses (in y and z directions), and
e. Combined stress, which is the sum of axial, bending-y and bending-z stresses.

All the stresses are calculated as the absolute value.

Related Links
G.18.1.4 Force Envelopes

Force envelopes of the member forces $F_X$ (axial force), $F_Y$ (Shear-y), and $M_Z$ (moment about local z-axis, i.e., strong axis) can be printed for any number of intermediate sections. The force values include maximum and minimum numbers representing maximum positive and maximum negative values.

The following is the sign convention for the maximum and minimum values:

- **$F_X$**
  A positive value is compression, and negative tension.

- **$F_Y$**
  A positive value is shear in the positive y-direction, and negative in the negative y-direction.

- **$F_Z$**
  Same as above, except in local z-direction.

- **$M_Z$**
  A positive moment will mean a moment causing tension at the top of the member. Conversely, a negative moment will cause tension at the bottom of the member. The top of a member is defined as the side towards positive local y-axis.

- **$M_Y$**
  Same as above, except about local y axis.

**Related Links**

- TR.43 Stress/Force Output Printing for Surface Entities (on page 2846)
- TR.45 Printing the Force Envelope (on page 2848)

G.19 Multiple Analyses

Structural analysis and design may require multiple analyses in the same run. STAAD.Pro allows you to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes.

For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an INACTIVE facility for this type of analysis.

**Inactive Members**

With the INACTIVE command, members can be made inactive. These inactive members will not be considered in the stiffness analysis or in any printout. The members made inactive by the INACTIVE command are made active again with the CHANGE command. This can be useful in an analysis where stage construction is modeled due to which, a set of members should be inactive for certain load cases. This can be accomplished by:

a. making the desired members inactive
b. providing the relevant load cases for which the members are inactive
c. performing the analysis
d. using the CHANGE command to make all the inactive members active
e. making another set of members inactive and providing the proper load cases for which the members are meant to be inactive, performing the analysis and repeating the procedure as necessary.

**Related Links**

- TR.38 Change Specification (on page 2835)
G.20 Steel, Concrete, and Timber Design

Related Links

- **D1.F. American Codes - Concrete Design per ACI 318** (on page 1478)
- **D. Steel Design** (on page 944)
- **D1.G. American Codes - Timber Design per AITC Code** (on page 1526)

G.21 Printing Facilities

All input data and output may be printed using PRINT commands available in STAAD. The input is normally echoed back in the output. However, if required, the echo can be switched off.

Extensive listing facilities are provided in almost all PRINT commands to allow you to specify joints, members and elements for which values are required.

Related Links

- **TR.44 Printing Section Displacements for Members** (on page 2846)
- **TR.45 Printing the Force Envelope** (on page 2848)
- **TR.46 Post Analysis Printer Plot Specifications** (on page 2849)

G.22 Miscellaneous Facilities

STAAD.Pro offers the following miscellaneous facilities for problem solution.

**Perform Rotation**

This command can be used to rotate the structure shape through any desired angle about any global axis. The rotated configuration can be used for further analysis and design. This command may be entered after the Joint Coordinates or between two Joint Coordinate commands or after all Member/Element Incidences are specified.

**Substitute**

Joint and member numbers may be redefined in STAAD through the use of the SUBSTITUTE command. After a new set of numbers are assigned, input and output values will be in accordance with the new numbering scheme. This facility allows the user to specify numbering schemes that will result in simple input specification as well as easy interpretation of data.
Calculation of Center of Gravity

STAAD is capable of calculating the center of gravity of the structure. The PRINT CG command may be utilized for this purpose.

Related Links

- TR.15 Redefinition of Joint and Member Numbers (on page 2439)
- TR.17 Rotation of Structure Geometry (on page 2444)
- TR.42 Print Specifications (on page 2840)
This section contains details of the STAAD.Pro commands used to create STAAD input files which are read by the STAAD engine.

**Note:** This section was previously included as Section 5 of the Technical Reference Manual. For convenience, the section numbers from that manual are maintained here with “TR.” in place of “5.” for the chapter number.

**Introduction**

The STAAD.Pro graphical user interface (GUI) is normally used to create all input specifications and all output reports and displays. These structural modeling and analysis input specifications are stored in STAAD input file—a text file with extension, .STD. When the GUI opens an existing model file, it reads all of the information necessary from the STAAD input file. You may edit or create this STAAD input file and then the GUI and the analysis engine will both reflect the changes.

The STAAD input file is processed by the STAAD analysis “engine” to produce results that are stored in several files (with file extensions such as ANL, BMD, TMH, etc.). The STAAD analysis text file (file extension .ANL) contains the printable output as created by the specifications in this manual. The other files contain the results (displacements, member/element forces, mode shapes, section forces/moments/displacements, etc.) that are used by the GUI in the post-processing mode.

**TR.0 STAAD Commands and Input Instructions**

This section of the manual describes in detail various commands and related instructions for STAAD.Pro. The user utilizes a command language format to communicate instructions to the program. Each of these commands either supplies some data to the program or instructs it to perform some calculations using the data already specified. The command language format and conventions are described in TR.1 Command Language Conventions (on page 2405). This is followed by a description of the available commands.

Although the STAAD.Pro input can be created through the Analytical Modeling workflow, it is important to understand the command language. With the knowledge of this language, it is easy to understand the problem and add or comment data as necessary. The general sequence in which the commands should appear in an input file should ideally follow the same sequence in which they are presented in this section. The commands are executed in the sequence entered. Obviously then the data needed for proper execution of a command must precede the command (e.g., Print results after Perform Analysis). Otherwise, the commands can be provided in any order with the following exceptions.

1. All design related data can be provided only after the analysis command.
2. All load cases and load combinations must be provided together, except in a case where the CHANGE command is used (see TR.38 Change Specification (on page 2835)). Additional load cases can be provided in the latter part of input.
All input data provided is stored by the program. Data can be added, deleted or modified within an existing data file.

In STAAD.Pro 2006 and earlier, all analytical calculations such as joint displacements, eigenvalues, and eigenvectors were calculated using an analysis engine which for identification purposes is known as the Basic Solver. This engine has been able to handle the analytical requirements for a vast majority of STAAD.Pro models that users have created in the recent years.

As computer resources such as processor speed, memory and disk space have grown, STAAD.Pro users are also creating larger models. As a result, numerically faster algorithms and solution techniques have become necessary. Also, new features such as pushover analysis and buckling analysis which are outside the scope of the standard engine have made it necessary to introduce a new engine which is known as the Advanced Solver. This new Solver is available as an alternative engine effective from STAAD.Pro 2007, Build 1001.

The Advanced Solver

As described above, the Advanced solver is a new addition to the STAAD Analysis Engine (note 1) which can be used for solving both static and dynamic problems. It is part of the STAAD.Pro engine with no special command required to run it. It is automatically activated if a suitable license is available (note 2), however, this can be turned off and the Basic Solver used by including the option:

```
SET STAR 0
```

This command must be included in the header information block at the start of the file and before the first JOINT command block.

The engine can operate in two modes, in-core and out-of-core. The in-core solver will be used for models with under 20000 joints and the out-of-core solver for models over 20000 joints. In most situations, the in-core mode will provide the quickest solution, but where there is insufficient memory available, then the engine will use the out-of-core mode. Again, selection of the mode is automatically chosen by the analysis, but can be over-ridden.

The full set of overrides for the advanced engine is:

```
SET STAR -3 use in-core solver regardless of size
SET STAR 4 use out-of-core solver regardless of size
SET STAR 3 default
SET STAR 0 use Basic STAAD solver
```

Notes

1. The Advanced Solver is *not* available for use with a Stardyne Analysis.
2. To use this feature requires access to a **STAAD Advanced license**. If you do not currently have this feature, please contact your account manager.
3. Global Euler Buckling analysis is different between the two solvers.

TR.1 Command Language Conventions

This section describes the command language used in STAAD. First, the various elements of the language are discussed and then the command format is described in details.
TR.1.1 Elements of STAAD Commands

**Integer Numbers**

Integer numbers are whole numbers written without a decimal point. These numbers are designated as \( i_1, i_2, \) etc., and should not contain any decimal point. Negative signs \((-)\) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number.

**Floating Point Numbers**

These are real numbers which may contain a decimal portion. These numbers are designated as \( f_1, f_2, \) etc. Values may have a decimal point and/or exponent. When specifying numbers with magnitude less than \( 1/100, \) it is advisable to use the E format to avoid precision related errors. Negative signs \((-)\) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number. Limit these to 24 characters.

Example

<table>
<thead>
<tr>
<th>5055.32</th>
<th>0.73</th>
<th>-8.9</th>
<th>732</th>
</tr>
</thead>
<tbody>
<tr>
<td>5E3</td>
<td>-3.4E-6</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

etc.

The decimal point may be omitted if the decimal portion of the number is zero.

**Alphanumeric**

These are characters, which are used to construct the names for data, titles or commands. Alphabetic characters may be input in upper or lower case letters. No quotation marks are needed to enclose them.

Example

MEMBER PROPERTIES
1 TO 8 TABLE ST W8X35

**Repetitive Data**

Repetitive numerical data may be provided in some (but not all) input tables such as joint coordinates by using the following format:

\[ n*f \]

Where:

- \( n = \) number of times data has to be repeated
- \( f = \) numeric data, integer and floating point

Example

JOINT COORDINATES
1 3*0.

This joint coordinate specification is same as:

1 0. 0. 0.
TR.1.2 Command Formats

Free-Format Input

All input to STAAD is in free-format style. Input data items should be separated by blank spaces (not commas) from the other input data items. Quotation marks are never needed to separate any alphabetic words such as data, commands or titles. Limit a data item to 24 characters.

Commenting Input

For documentation of a STAAD data file, the facility to provide comments is available. Comments can be included by providing an asterisk (*) mark as the first non-blank character in any line. The line with the comment is "echoed" in the output file but not processed by the program.

Example

JOINT LOAD
* THE FOLLOWING IS AN EQUIPMENT LOAD
2 3 7 FY 35.0
* etc.

Meaning of Underlining in the Manual

Exact command formats are described in the latter part of this section. Many words in the commands and data may be abbreviated. The full word intended is given in the command description with the portion actually required (the abbreviation) underlined.

For example, if the word MEMBER is used in a command, only the portion MEB need be input. It is clearer for others reading the output if the entire word is used, but an experienced user may desire to use the abbreviations.

Meaning of Braces and Parenthesis

In some command formats, braces enclose a number of choices, which are arranged vertically or separated by a | character. One and only one of the choices can be selected. However, several of the listed choices may be selected if an asterisk (*) mark is located outside the braces.

Example

{XY | YZ | XZ}

In the above example, you must make a choice of XY or YZ or XZ.

Note: In some instances, the choices will be explicitly defined using "or" for clarification.

Example

*{FX | FY | FZ}

Here, you can choose one or all of the listing (FX, FY and FZ), in any order.

Parentheses, (), enclosing a portion of a command indicate that the enclosed portion is optional. The presence or absence of this portion affects the meaning of the command, as is explained in the description of the particular command.
Example
PRINT (MEMBER) FORCES
PERFORM ANALYSIS (PRINT LOAD DATA)

In the first line, the word MEMBER may be omitted with no change of the meaning of the command. In the second line, the PRINT LOAD DATA command may also be omitted, in which case the load data will not be printed.

Multiple Data Separator
Multiple data can be provided on a single line, if they are separated by a semicolon (;) character. One restriction is that a semicolon can not separate consecutive commands. They must appear on separate lines.

Example
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4
etc.

Possible Error:
PRINT FORCES; PRINT STRESSES
In the above case, only the PRINT FORCES command is processed and the PRINT STRESSES command is ignored.

Listing Data
In some STAAD command descriptions, the word "list" is used to identify a list of joints, members/elements, or loading cases. The format of a list can be defined as follows:
list = *( i_1 i_2 i_3 ... | i_1 TO i_2 (BY i_3) | X or Y or Z )

TO means all integers from the first (i_1) to the second (i_2) inclusive. BY means that the numbers are incremented by an amount equal to the third data item (i_3). If BY i_3 is omitted, the increment will be set to one. Sometimes the list may be too long to fit on one line, in which case the list may be continued to the next line by providing a hyphen preceded by a blank. Also, only a list may be continued and not any other type of data.

Instead of a numerical list, a single group-name may be entered if that group was previously defined.

Instead of a numerical list, the specification X (or Y or Z) may be used. This specification will include all MEMBERS parallel to the global direction specified. Note that this is not applicable to JOINTs or ELEMENTs.

Note: ALL, BEAM, PLATE, SOLID. Do not use these unless the documentation for a command specifically mentions them as available for that command. ALL means all members and elements, BEAM means all members, etc.

Continuing a command to the next line
Only lists may be continued to the next line by ending the line with a blank and hyphen (see above) with few exceptions: Multilinear spring supports, Supports, Master/Slave. Others have special types of continuations. Please follow the command descriptions.

Example
2 4 7 TO 13 BY 2 19 TO 22 -
28 31 TO 33 FX 10.0
This list of items is the same as:

```
2 4 7 9 11 13 19 20 21 22 28 31 32 33 FX 10.0
```

Possible Error:

```
3 5 TO 9 11 15 -
FX 10.0
```

In this case, the continuation mark for list items is used when list items are not continued. This will result in an error message or possibly unpredictable results.

**TR.1.3 Listing of Objects by Specification of Global Ranges**

Used to specify lists of objects (e.g., joints, members, and/or elements) by providing global ranges. The general format of the specification is as follows.

**General Format**

```
{ XRANGE | YRANGE | ZRANGE } f1, f2
```

Where:

- XRANGE, YRANGE, ZRANGE = direction of range (parallel to global X, Y, Z directions respectively)
- f1, f2 = values (in current unit system) that defines the specified range.

**Notes**

1. Only one range direction (XRANGE, YRANGE etc.) is allowed per list. (Exceptions: Area/Floor load and Master/Slave).
2. No other items may be in the list.
3. The values defining the range (f1, f2) must be in the current unit system.

**Example**

```
MEMBER TRUSS
XRANGE 20. 70.
CONSTANTS
E STEEL YRANGE 10. 55.
```

In the above example, a XRANGE is specified with values of 20. and 70. This range will include all members lying entirely within a range parallel to the global X-axis and limited by X=20 and X=70.

**TR.2 Problem Initiation and Model Title**

This command initiates the STAAD run and is also used to specify the type of the structure and provide an optional title.

Any STAAD input file must start with the word STAAD. Following type specifications are available:

- **PLANE**= Plane frame structure
- **SPACE**= Space frame structure
- TRUSS= Plane or space truss structure
- FLOOR= Floor structure

**General Format**

\[
\text{STAAD \{ PLANE | SPACE | TRUSS | FLOOR \} (any\_title)}
\]

Where:

\[\text{any\_title} = \text{Any title for the problem. This title will appear on the top of every output page. To include additional information in the page header, use a comment line containing the pertinent information as the second line of input.}\]

**Notes**

1. Care must be taken about choosing the type of the structure. The choice is dependent on the various degrees of freedom that need to be considered in the analysis. The following figure illustrates the degrees of freedoms considered in the various type specifications. Detailed discussions are available in [G.2 Types of Structures](on page 2294). PLANE indicates the XY plane for Y up and the XZ plane for Z up. FLOOR indicates the XZ floor for Y up and the XY floor for Z up.

   ![Figure 265: Structure type A) Plane, B) Space, C) Truss 2D or 3D, and D) Floor](image)

2. The optional title provided by you are printed on top of every page of the output. You can use this facility to customize his output.

**Limits**

The following limits to model size are effective for STAAD.Pro CONNECT Edition (release 21.00.00) and later.

- Joint number 1 to 999,999
- Number of joints: 200,000*
- Member/Element numbers: 1 to 999,999
- Number of Members, Plates, and Solids: 225,000*
Load Case numbers: 1 to 99,999
Number of primary and combination cases: 10,101
Number of modes and frequencies: 2,700
Number of load cases that may be combined by a Repeat Load or Load Combination command: 550

* Some STAAD copies are available with much smaller limits, please check what limits you have purchased.

Notes:
The numerical limits should be considered as upper limits built into the software for those quantities on an individual basis. In practice, the actual maximums the program can handle are determined by the hardware resources as well as the limits imposed by the operating system. For example, it is highly improbable that a single model with 999,999 members and 99,999 load cases can be solved.

The memory demand of the program is determined by the combined effect of two or more of these terms. For example, when a steel design is performed, the memory required depends on the product of the members being designed (NMD) as well as the number of load cases being designed for (NL). That is, NMD ×NL. So the smaller the NMD, the larger the NL capacity and vice versa.

Experience has shown that for a medium size model of 5,000 frame members, about 1,500 load cases is the practical limit of what can be run for the 32-bit program on a computer with 16 GB RAM and 200GB of free disk space.

Related Links
• G.2 Types of Structures (on page 2294)

TR.3 Unit Specification
This command allows you to specify or change length and force units for input and output.

Description
The magnitude of numerical data that is entered in a STAAD model is determined by the unit of that data.
STAAD.Pro supports numerical data based on both English and Metric unit systems.

The method used by STAAD.Pro to determine the unit of any value is by the previous UNIT command.

The UNIT command can set or change the unit of LENGTH, FORCE, or both. This can be set any number of times in a STAAD input file subject to the conditions noted below.

General Format

UNIT *{ length-unit | force-unit }

where:

length-unit = { INCHES | FT or FEET | METER | CM | DME | MMS | KM }

force-unit = { KIP | POUND | KNS | DNS | NEWTON | MNS | MTON | KG }

The following tables illustrate the unit and the relative factor against the primary base unit:
Table 226: Units of length in STAAD.Pro

<table>
<thead>
<tr>
<th>Required Length Unit</th>
<th>Base Unit</th>
<th>Factor</th>
<th>UNIT command parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>English</td>
<td>Inch</td>
<td>1</td>
<td>IN</td>
</tr>
<tr>
<td></td>
<td>Foot</td>
<td>12</td>
<td>FT (or FE)</td>
</tr>
<tr>
<td>Metric</td>
<td>Meter</td>
<td>1</td>
<td>ME</td>
</tr>
<tr>
<td></td>
<td>Decimeter</td>
<td>0.1</td>
<td>DM</td>
</tr>
<tr>
<td></td>
<td>Centimeter</td>
<td>0.01</td>
<td>CM</td>
</tr>
<tr>
<td></td>
<td>Millimeter</td>
<td>0.001</td>
<td>MM</td>
</tr>
<tr>
<td></td>
<td>Kilometer</td>
<td>1,000</td>
<td>KM</td>
</tr>
</tbody>
</table>

Table 227: Units of force in STAAD.Pro

<table>
<thead>
<tr>
<th>Required Force Unit</th>
<th>Base Unit</th>
<th>Factor</th>
<th>UNIT command parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>English</td>
<td>Kip (kilopound)</td>
<td>1</td>
<td>KI</td>
</tr>
<tr>
<td></td>
<td>Pound</td>
<td>0.001</td>
<td>PO</td>
</tr>
<tr>
<td>Metric</td>
<td>Kilonewton</td>
<td>1</td>
<td>KN</td>
</tr>
<tr>
<td></td>
<td>Decanewton</td>
<td>0.1</td>
<td>DN</td>
</tr>
<tr>
<td></td>
<td>Newton</td>
<td>0.001</td>
<td>NE</td>
</tr>
<tr>
<td></td>
<td>Meganewton</td>
<td>1,000</td>
<td>MN</td>
</tr>
<tr>
<td></td>
<td>Metric Ton</td>
<td>9.80665</td>
<td>MT</td>
</tr>
<tr>
<td></td>
<td>Kilogram</td>
<td>9.80665 (10)^{-3}</td>
<td>KG</td>
</tr>
</tbody>
</table>

Example

UNIT KIP FT
UNIT INCH
UNIT METER KNS
UNIT CM MTON
Notes

a. A UNIT command may be inserted before any primary level command (e.g., JOINT COORD, MEMBER INCIDENCE, etc.).

b. If no UNIT command is specified, then the default units are taken as:
   - Length: feet
   - Force: kips

c. Units can be mixed between systems of units, such as length of feet and force of kilonewtons.

d. Units that include both length and force components (e.g., stress = force/length^2) will be constructed from the current length and force units.

   Thus, if the current length unit is feet and force unit is kilonewtons, then when defining Young’s modulus, the value for a stress unit is kN/ft^2. While this may be an odd definition, it is fully supported by STAAD.Pro.

e. Kilograms and metric tons are mass units which are converted to units of force assuming an acceleration due to gravity of 32.174 ft/s^2.

f. Temperature units are not explicitly defined in STAAD.Pro. The value of temperature loading and coefficient of thermal expansion are taken to be consistent units.

Related Links

- G.3 Unit Systems (on page 2295)

TR.4 Input/Output Width Specification

These commands may be used to specify the width(s) of the lines of output file(s).

For INPUT width, 79 is always used. The program can create output using two different output widths - 72 (default) and 118. The narrower, 72-character width is used for display on most older monitors and for printing on “portrait” wide paper. The 118-character width may be used for printing on “landscape” wide paper.

Note: This is a customization facility that may be used to improve the presentation quality of the output documents.

General Format

{INPUT | OUTPUT} WIDTH i1

Where:

i1 = 72 or 118 depending on narrow or wide output. 72 is the default value.

TR.5 Set Command Specification

This command allows you to set various general specifications for the analysis/design run.
General Format

```
SET { NL i1 | {DISPLACEMENT i2 | PDELTA TOLERANCE i9} | SDAMP i3 | WARP i4 | ITERLIM i5 | PRINT i7 | NOPRINT DIRECT | SHEAR | ECHO { ON | OFF } | GUI i6 | { Z | Y | UP } | DEFLECTION CUTOFF f1 | FLOOR LOAD TOLERANCE f2 | EIGEN METHOD { LANCZOS | RITZ} }
```

Where:

Description

The following SET commands contain values with associated units and should appear after a UNIT command and before the first JOINT command.

Table 228: Commonly used SET commands which take units

<table>
<thead>
<tr>
<th>Command</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET NL</td>
<td>i1</td>
<td>The SET NL command is used in a multiple analysis run if you want to add more primary load cases after one analysis has been performed. Specifically, for those examples which use the CHANGE command (see TR.38 Change Specification (on page 2835)), if you want to add more primary load cases, the NL value should be set to the maximum number (or slightly more) with the SET NL command. The program will then be able to set aside additional memory space for information to be added later. This command should be provided before any joint, member or load specifications. The value for i1 is the maximum number of primary load cases (NL). This value should not be much greater than the maximum number of primary load cases actually used in the model.</td>
</tr>
</tbody>
</table>
| SET DISPLACEMENT   | i2        | For PDELTA ANALYSIS with CONVERGE option there are two convergence methodologies to choose from (refer to TR.37.2 P-Delta Analysis Options (on page 2797) for additional information)  
1. The SET DISPLACEMENT i2 command is used to specify the convergence tolerance. If the Euclidean norm RMS displacement of two consecutive iterations changes less than the value entered, then that load case is converged. This command should be placed before the JOINT COORDINATE specification. The default tolerance value, i2, is equal to the maximum span of structure divided by 120. The convergence tolerance for the Euclidean norm is difficult to know, so using this option is not recommended.  
2. The SET PDELTA TOL i9 command selects this second method. Default tolerance value, i9, is 0.01 inch. If the maximum change in each displacement dof from two consecutive iterations is less than ftol, then that load case is... |

STAAD.Pro 2414 User Manual
### Command

<table>
<thead>
<tr>
<th>Command</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PDELTA TOLERANCE</td>
<td>i9</td>
<td>converged. This command should be placed before the JOINT COORDINATE specification and after a UNIT command.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Both i2 and i9 are the P-Delta Convergence criteria input. There are two methods available, enter SET DISPLACEMENT i2 or SET PDELTA TOL i9. Enter only one.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( i2 = ) If the change in the Euclidean norm of the displacement vector from one PDELTA iteration to the next is less than this convergence tolerance value; then the iteration has converged for the case being analyzed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( i9 = ) If the maximum absolute change in displacement of each dof from one PDELTA iteration to the next is less than this convergence tolerance value; then the iteration has converged for the case being analyzed.</td>
</tr>
<tr>
<td>SET DEFORMATION CUTOFF</td>
<td>f1</td>
<td>Used to arrest huge displacements in minor axis due to small delta effects. ( f1 ) = If the absolute value of the maximum section displacement is less than ( f1 ) after two iterations; then it is converged. Rapidly diverging minor axis displacement will not occur until after two iterations. ( f1 ) is in current length units.</td>
</tr>
<tr>
<td>SET FLOOR LOAD TOLERANCE</td>
<td>f2</td>
<td>Used to specify the tolerance for out of plane nodes in a floor load. The value is given as a length, in current units. The default is taken as 0.01% of the length of the longest beam of the beams in the floor load command. For example, if a floor group is used, then beams in the floor group are used in the computation. Similarly, when a Y range is used, beams within that range are used. Refer to TR.32.4.3 Floor Load Specification (on page 2672) for details on specifying a plane for floor loads. ( f2 ) = tolerance value used</td>
</tr>
</tbody>
</table>

The following SET commands have dimensionless input
<table>
<thead>
<tr>
<th>Command</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET SDAMP</td>
<td>i3</td>
<td>The SET SDAMP command will allow the damping of springs to be considered in computing the composite modal damping for each mode in a dynamic solution. This command is not used unless CDAMP ratios are also entered for the members and elements in the CONSTANTS command. Composite damping is generally only used if there are many modes in the dynamic solution and there are a wide range of damping ratios in the springs, members, or elements. i3 = the damping ratio to be used for all springs in computing the modal composite damping in dynamics.</td>
</tr>
<tr>
<td>SET WARP</td>
<td>i4</td>
<td>The SET WARP command will allow the I section member end warping restraint to be considered in calculating the torsional stiffness rigidity. Full or partial or no warping restraint are allowed.</td>
</tr>
<tr>
<td>SET ITERLIM</td>
<td>i5</td>
<td>The SET ITERLIM command is for raising the maximum iteration limit above the default of 10 in tension/compression iterations. Since this iterative procedure will not necessarily converge, this option of more iterations may not help and should be used with caution. The minimum iteration limit that may be entered is 3. After any tension/compression analysis, the output file (file extension .ANL) should be scanned for warnings of non-convergence. Do not use results from non-converged cases. i5 = Maximum number of tension/compression iterations.</td>
</tr>
</tbody>
</table>
| SET PRINT   | i7        | The following values can be used to suppress the described warnings or include the described additional results in the output:  

1 = Omit zero stiffness message, Rotational zero stiffness message due to solids, and "Node not connected. OK if master/slave" message.  
2 = Omit Member in list does not exist message.  
3 = Omit joint not connected message.  
5 = Turn off floor load message.  
8 = Omit self weight warnings.  
10 = Turns on some iteration messages in direct analysis.  
17 = Write rotational masses to mass text file; otherwise only translational masses written. Print scaled modal results for RSA and some force data by floor for RSA.  

i7 = Used to suppress some warning messages or to include additional output. |
<table>
<thead>
<tr>
<th>Command</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET NOPRINT</td>
<td>(n/a)</td>
<td>Used to turn off the tau-b details in the output file when running a Direct Analysis (refer to TR.37.5 Direct Analysis (on page 2806)).</td>
</tr>
<tr>
<td>DIRECT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SET SHEAR</td>
<td>(n/a)</td>
<td>The SET SHEAR command is for omitting the additional pure shear distortion stiffness terms in forming beam member stiffnesses. With this command you can exactly match simple textbook beam theory results.</td>
</tr>
<tr>
<td>SET ECHO</td>
<td>(n/a)</td>
<td>The SET ECHO ON command will activate and the SET ECHO OFF command will deactivate the echoing of input file commands in the output file. In the absence of the SET ECHO command, input file commands will be echoed back to the output file.</td>
</tr>
</tbody>
</table>
| SET GUI       | i6        | After the calculations are completed, and before the Analysis window is closed, the program creates several files for the purpose of displaying results in the post processing mode. In large models, this can be a time consuming process. If the user's goal is to look at results in the output file only (file extension .ANL) and does not intend to go into the post-processing mode, he/she could instruct the program to skip the process of creating those files. The SET GUI 1 may be specified immediately before the FINISH command, or somewhere near the beginning of the file after STAAD SPACE.  
   i6 = 1, Bypass forming data and files needed for post-processing. |
<table>
<thead>
<tr>
<th>Command</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET {Z</td>
<td>Y} UP</td>
<td>(n/a)</td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image" alt="Diagram of global axis orientation options in STAAD.Pro" /></td>
</tr>
</tbody>
</table>

Note: Both options follow the “right hand rule.”

Figure 266: Global axis orientation options in STAAD.Pro

The SET Z UP Command directly influences the values of the following input:

a. JOINT COORDINATE
b. Input for the PERFORM ROTATION Command
c. BETA ANGLE

Note: The following features of STAAD cannot be used with the SET Z UP command:

- Automatic Generation of Spring Supports for Mat Foundations

---

<table>
<thead>
<tr>
<th>Command</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET STAR</td>
<td>i</td>
<td>Instructs the program which solver to use. Refer to TR.0 STAAD Commands and Input Instructions (on page 2404) for additional information on using the Advanced Solver.</td>
</tr>
</tbody>
</table>

0 = STAAD Basic Solver  
1 = Other solvers  
2 = STAAD Advanced Solver (in-core)  
3 = STAAD Advanced Solver (out-of-core)
### Command Specification

<table>
<thead>
<tr>
<th>Command</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
</table>
| SET EIGEN METHOD \{ LANCZOS | RITZ \} | (n/a) | **LANCZOS** — Instructs the solver to use Arnoldi/Lanczos method for extraction of eigen vectors.  
**Note:** SET EIGEN METHOD is not supported by the basic solver. If the command is present, a warning message is given in the output.  
**RITZ** — Use load dependent Ritz vectors method for extraction of eigen vectors.  
If neither method is specified, the default method of Subspace-iteration is used.  
**Note:** SET EIGEN METHOD RITZ is not supported by pushover analysis or steady state analysis. In the case of a pushover analysis, this is because the lateral load distribution is based on the eigen vector. If the command is present, a warning message is given in the output. |

### Less frequently used SET commands

The following table contains a list of less frequently used SET commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET SOLUTION INCORE</td>
<td>In Basic solver use determinant search method for frequencies for small problems.</td>
</tr>
<tr>
<td>SET INTERPOLATION { LIN</td>
<td>LOG }</td>
</tr>
<tr>
<td>SET NOSECT</td>
<td>No section results will be calculated</td>
</tr>
</tbody>
</table>
| SET PLATE FLATNESS TOLERANCE $f$ | Used to so specify a tolerance for the plate warping check. By default, plate warping is reported when there is a 30 degree angle (or larger) between the normal to two triangles which are formed by splitting a quad element. This splitting can be done two ways so there are four triangles. If any have more than the tolerance angle (in degrees) between them, then an error is reported.  
The value $f$ is given in degrees. |
<p>| SET INPLANE ROTATION | In-plane rotation (MX) in plates will be ignored. |
| SET NANGLE | |
| SET FOOTER | Old or new procedure |</p>
<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET RESTART i</td>
<td>Basic solver only:</td>
</tr>
<tr>
<td></td>
<td>0 = Create solved matrix as scratch (default)</td>
</tr>
<tr>
<td></td>
<td>1 = Do not delete after creating matrix</td>
</tr>
<tr>
<td></td>
<td>2 = Use existing solved matrix</td>
</tr>
<tr>
<td>SET ENDFACTOR f</td>
<td>1.0 or -1.0 for combining spectrum cases</td>
</tr>
<tr>
<td>SET ORTHOG</td>
<td>1 = Orthogonalize trial vectors in SSPCEB</td>
</tr>
<tr>
<td>SET PROFILE s2</td>
<td>Specify the folder path to read the section property (.mdb) files</td>
</tr>
<tr>
<td>SET NOWARNING</td>
<td>Switches off some warning messages.</td>
</tr>
<tr>
<td>SET USE DLL</td>
<td>Used plugin .dll for design (ACI code only)</td>
</tr>
<tr>
<td>SET RIGID DIAPHRAGM i</td>
<td>Number of rigid diaphragms.</td>
</tr>
<tr>
<td>SET SSVECT</td>
<td>Basic solver only</td>
</tr>
<tr>
<td></td>
<td>To instruct the program to use a different initial set of trial vectors for eigensolution. May be used if eigen extraction fails.</td>
</tr>
<tr>
<td>SET INCLINED REACTION</td>
<td>To obtain reactions at inclined supports in the inclined axis system.</td>
</tr>
<tr>
<td>SET GROUP DUPLICATES</td>
<td>Followed by an integer value which is used to specify the number of groups to which one model entity (node, member, plate, or solid object) may belong. Must be in the range of four to 100, inclusive. The default value is 10 groups.</td>
</tr>
<tr>
<td>SET CG TXT</td>
<td>Used to print section forces for generated load combinations to an external text file. See TR.35 Load Combination Specification (on page 2791) for additional details. This command should be used only for review of data and not in normal circumstances as the resulting text file can be very large.</td>
</tr>
<tr>
<td>SET RS TXT</td>
<td>Used to print section forces for each of the modes for response spectrum load cases to an external text file. See TR.35 Load Combination Specification (on page 2791) for additional details. This command should be used only for review of data and not in normal circumstances as the resulting text file can be very large.</td>
</tr>
<tr>
<td>Command</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>SET NF TXT</td>
<td>Used to print section forces for each member corresponding to the critical load combination generation case to an external text file. See TR.35 Load Combination Specification (on page 2791) for additional details. This command should be used only for review of data and not in normal circumstances as the resulting text file can be very large.</td>
</tr>
<tr>
<td>SET PRINT STIFFNESS</td>
<td>Prints the assembled global stiffness matrix to the output.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> This command is only applicable to the Basic solver. It will be ignored if the STAAD Advanced Solver is used. (You can use in conjunction with SET STAR 0 to enforce the basic solver)</td>
</tr>
<tr>
<td></td>
<td>The output includes the stiffness matrix from the diagonal and only includes non-zero terms (with values of $10^{-20}$ or less are assumed to be zero). See the section below on how to read this output.</td>
</tr>
<tr>
<td></td>
<td>See Stiffness Matrix Output (on page 2422) for details on how to interpret the stiffness matrix output.</td>
</tr>
<tr>
<td>SET THCOPYS i</td>
<td>i = 4 (default)</td>
</tr>
<tr>
<td></td>
<td>i = number of TYPES that a given node-dof may be a part of in time history loading.</td>
</tr>
<tr>
<td>SET WARNING { ON</td>
<td>OFF }</td>
</tr>
<tr>
<td>SET NOTE { ON</td>
<td>OFF }</td>
</tr>
<tr>
<td>SET DIVISION i</td>
<td>Set the number of divisions used in meshing for surfaces (default is 10)</td>
</tr>
<tr>
<td>SET MULTI { 1</td>
<td>2 }</td>
</tr>
<tr>
<td>SET PARTICIPATION FACTOR</td>
<td>Compute participation factors as a numeric value in addition to the percentage of total mass usually displayed.</td>
</tr>
<tr>
<td>SET BUCKLING MODES i</td>
<td>Number of buckling modes computed with advanced solver. Default is i = 4.</td>
</tr>
</tbody>
</table>
### Command

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>
| SET BYPASS { DIS | FOR | EJS | EJF }               | Bypass generating certain outputs for the graphical user interface.  
DIS = section displacements  
FOR = section force  
EJS = plate nodal stress  
EJF = plate nodal force |
| SET LOAD PLATE           | Include loads applied to inactive plates.        |

### Stiffness Matrix Output

When the `SET PRINT STIFFNESS` command is used with the Basic solver, the non-zero terms for the diagonal and upper half of the stiffness matrix are reported. The following example shows the output of adding this command to the file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Static Beams\BEAM01.std`

```
NON-ZERO STIFFNESS MATRIX VALUES, PRINTED BY ROWS FROM THE DIAGONAL.

<table>
<thead>
<tr>
<th>ROW</th>
<th>JOINT</th>
<th>DIRECTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>6</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>2</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>6</td>
</tr>
</tbody>
</table>
```

This describes position of each non-zero term in a row starting from the diagonal and then moving right. So, this matrix would be written as:

\[
\begin{bmatrix}
118, 009.9515 & 0 & -1, 498.726372 & 59, 004.97573 \\
3, 172.303322 & 0 & 0 & sym.
\end{bmatrix}
\begin{bmatrix}
32.89766633 & -832.6259057 \\
196, 683.244
\end{bmatrix}
\]

### Related Links

- [G.17.3.1 Solution of the Eigenproblem](on page 2363)

### TR.6 Data Separator

This command may be used to specify the desired separator character that can be used to separate multiple lines of data on a single line of input.

The semicolon (;) is the default character which functions as the separator for multiple line data on one line. However, this separator character can be changed by the `SEPARATOR` command to any character.

**Note:** Comma (,) or asterisk (*) may not be used as a separator character.
Technical Reference of STAAD Commands

TR.7 Page Control Commands

TR.7.1 Page New

This command may be used to instruct the program to start a new page of output.

With this command, a new page of output can be started. This command provides the flexibility, the user needs, to design the output format.

Note: The presentation quality of the output document may be improved by using this command properly.

General Format

```
SEPARATOR a
```

TR.7.2 Page Length

This command may be used to specify the page length of the output.

General Format

```
PAGE LENGTH i
```

Where:

- **i**: The page length in STAAD output is based on a default value of 60 lines. However, you may change the page length to any number \( i \) (number of lines per page) wanted.

TR.8 Ignore Specifications

This command allows you to provide member lists in a convenient way without triggering error messages pertaining to non-existent member numbers.

The **IGNORE LIST** command may be used if you want the program to ignore any nonexistent member(s) that may be included in a member list specification. For example, for the sake of simplicity, a list of members may be specified as MEMB 3 TO 40 where members 10 and 11 do not exist. An error message can be avoided in this situation by providing the **IGNORE LIST** command anywhere in the beginning of input. A warning message, however, will appear for each nonexistent member.

General format

```
IGNORE LIST
```
TR.9 No Design Specification

This command allows you to declare that no design operations will be performed during the run. The memory reserved for design will be released to accommodate larger analysis jobs.

STAAD always assumes that at some point in the input, you may want to perform design for steel, concrete, etc. members. These design processes require more computer memory. If memory availability is a problem, the above command may be used to eliminate extra memory requirements.

General Format

```
INPUT NODESIGN
```

TR.10 Job Information Data

This optional block of commands is used to project metadata for the STAAD input file.

General Format

The job information data block must begin with the START and END lines, with all lines in between being optional.

```
START JOB INFORMATION

ENGINEER DATA date
JOB NAME job-name
JOB CLIENT client-name
JOB NO job-number
JOB REV revision-number
JOB PART part-number
JOB REF reference-number
JOB COMMENT comments-string
ENGINEER NAME engineer-name
CHECKER NAME checked-by-name
APPROVED NAME approved-by-name
CHECKER-DATE checked-on-date
APPROVED DATE approved-on-date
CONNECTED PROJECT ID project-id
CONNECTED PROJECT NAME project-name
END JOB INFORMATION
```
**Note:** The job information block must be placed before the `JOINT COORDINATES` command.

**Caution:** The CONNECTED Project `project-id` and `project-name` should not be directly edited. These are added through the User Interface.

---

**TR.11 Joint Coordinates Specification**

These commands allow you to specify and generate the coordinates of the joints of the structure. The `JOINT COORDINATES` command initiates the specification of the coordinates. The `REPEAT` and `REPEAT ALL` commands allow easy generation of coordinates using repetitive patterns.

### General Format

```plaintext
JOINT COORDINATES (CYLINDRICAL (REVERSE)) (NOCHECK) (NOREDUCE BAND)

n xi1 yi1 zi1 (xi2 yi2 zi2 ... xin yin zin)

REPEAT n xi1 yi1 zi1 (xi2 yi2 zi2 ... xin yin zin)

REPEAT ALL n xi1 yi1 zi1 (xi2 yi2 zi2 ... xin yin zin)
```

- `n` is limited to 150
- `JTORIGIN xOrigin yOrigin zOrigin`

Where:

- `i1` = the reference number of the node coordinate that follows
- `x1 y1 z1` = the coordinates of node `i1` in the given coordinate system
- `i2` = optional last node number of a sequence of nodes generated from node `i1`
- `x2 y2 z2` = the coordinates of node `i2` in the given coordinate system
- `i3` = the increment in node number from `i1` to `i2`. Default is 1. Each node generated will be equally spaced between `i1` and `i2`.

This line is repeated to define each node or each generated set of nodes.

A line starting with `REPEAT` will generate `n` copies of the previous line with an offset given by `xi1 yi1 zi1`.

A line starting with `REPEAT ALL` will generate `n` copies of all the previous lines following any `REPEAT ALL` specification with an offset given by `xi1 yi1 zi1`.

The optional parameter `JTORIGIN` causes the program to use a different origin than (0,0,0) for all joints entered with this `JOINT COORDINATES` command. It is useful in instances such as when the center of a cylinder is not at (0,0,0) but at a different point in space. The `JTORIGIN` command should be entered on a separate line. After the joint coordinates are entered or generated, then the `xOrigin`, `yOrigin`, `zOrigin` values are added to the respective X, Y, and Z coordinates of each joint.

The values of the coordinates (x y z) for cylindrical and reverse cylindrical systems are determined as follows:

The cylindrical coordinate system defines a point in space by virtue of the radius, an angle, and an offset distance:

1. `x1` = the radial distance (in current length units) from the origin in the XY plane
2. `y1` = an angle (in degrees) measured from the global X axis counter-clockwise about the global Z axis
3. \( z_1 \) = the offset distance (in current length units) from the XY plane along the global Z axis

\[ R \theta \]

\[ X \]

\[ Y \]

\[ Z \]

Figure 267: Cylindrical Coordinate System

The reference cylindrical coordinate system is an alternative cylindrical system that defines a point in space by virtue of the radius, an offset distance, and an angle:

1. \( x_1 \) = the radial distance (in current length units) from the origin in the XZ plane
2. \( y_1 \) = the offset distance (in current length units) from the XZ plane along the global Y axis
3. \( z_1 \) = an angle (in degrees) measured from the global X axis counter-clockwise about the global Y axis

\[ R \theta \]

\[ X \]

\[ Y \]

\[ Z \]

Figure 268: Reverse Cylindrical Coordinate System

Description

The command JOINT COORDINATES specifies a Cartesian coordinate system. Joints are defined using the global X, Y, and Z coordinates. The command JOINT COORDINATES CYLINDRICAL specifies a cylindrical coordinate system. Joints are defined using the \( r \), \( q \), and \( z \) coordinates. JOINT COORDINATES CYLINDRICAL REVERSE specifies a reverse cylindrical coordinate system. Joints are defined using the \( r \), \( y \), and \( q \) coordinates. Refer to G. 4.1 Global Coordinate System (on page 2296) for details and figures.

The multiple JOINT COORDINATES command concept allows UNIT changes and PERFORM ROTATION commands in between; such that these commands would apply to a selected portion of the joints. However, the PERFORM
ROTATION command applies to all prior defined joints, not just those in the previous JOINT COORDINATE command.

Example 1

JOINT COORDINATES
1 0 0 0 3 3 0 0 1

Generates the following three nodes:

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>1.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>3.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Example 2

JOINT COORDINATES
1 0 0 0 5 5 0 0 2

Generates the following three nodes:

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>2.5</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>5.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Example 3

JOINT COORDINATES CYLINDRICAL
1 10.0 0.0 0.0 5 10.0 180 0.0 1

Generated the following five nodes:

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>7.1</td>
<td>7.1</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>0.0</td>
<td>10.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>-7.1</td>
<td>7.1</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>-10.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>
The following examples illustrate various uses of the REPEAT command.

Example 4

<p>| | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>0.0</td>
<td>10.0</td>
<td>0.0</td>
<td>;</td>
<td>22</td>
<td>15.0</td>
<td>10.0</td>
</tr>
<tr>
<td>40</td>
<td>45.0</td>
<td>10.0</td>
<td>60.0</td>
<td>;</td>
<td>41</td>
<td>0.0</td>
<td>20.0</td>
</tr>
<tr>
<td>200</td>
<td>45.0</td>
<td>90.0</td>
<td>60.0</td>
<td>;</td>
<td>201</td>
<td>0.0</td>
<td>100.0</td>
</tr>
<tr>
<td>219</td>
<td>30.0</td>
<td>100.0</td>
<td>60.0</td>
<td>;</td>
<td>220</td>
<td>45.0</td>
<td>100.0</td>
</tr>
</tbody>
</table>

REPEAT 10  5. 10. 5.

The above REPEAT command will repeat the last input line 10 times using the same set of increments (i.e., x = 5., y = 10., z = 5.)

REPEAT 3  2. 10. 5. 3. 15. 3. 5. 20. 3.

The above REPEAT command will repeat the last input line three times. Each repeat operation will use a different increment set.

REPEAT 10  0. 12. 0. 15*0 0. 10. 0. 9*0

The above REPEAT command will repeat the last input line 10 times; six times using x, y and z increments of 0., 12. and 0., and four times using increments of 0., 10. and 0. Each x, y, and z value of 0 represents no change from the previous increment. To create the 2nd through 6th repeats, five sets of 0., 0. and 0. (15*0) are supplied. The seventh repeat is done with increments of 0., 10. and 0. The 8th through 10th repeats are done with the same increments as 7, and is represented as 9*0.

**Note:** The PRINT JOINT COORDINATE command may be used to verify the joint coordinates provided or generated by REPEAT and REPEAT ALL commands. Also, you can use the Postprocessing workflow to verify geometry graphically.

**Related Links**

- G.4 Coordinate Systems and Structure Geometry (on page 2295)
- G.4.1 Global Coordinate System (on page 2296)
- G.4.2 Local Coordinate System (on page 2297)
- G.4.3 Relationship Between Global and Local Coordinates (on page 2301)
- G.4.2 Local Coordinate System (on page 2297)
- G.5 Finite Element Information (on page 2308)
- G.5.1 Plate and Shell Elements (on page 2308)
- G.5.2 Solid Elements (on page 2319)
- G.5.3 Surface Elements (Deprecated) (on page 2322)

**TR.12 Member Incidences Specification**

This set of commands is used to specify members by defining connectivity between joints. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The member/element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD.Pro is capable of detecting multiple structures automatically (on page 887).
General Format

MEMBER  INCIDENCES

\[ i_1 \ i_2 \ i_3 \ ( \ i_4 \ i_5 \ i_6 \ ) \]

REPEAT  \[ n \ \ m_1 \ j_1 \]

REPEAT  ALL  \[ n \ \ m_1 \ j_1 \]

Description

The REPEAT command causes the previous line of input to be repeated \( n \) number of times with specified member and joint increments. The REPEAT  ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT  ALL command or to the beginning of the specification if no previous REPEAT  ALL command has been issued.

**Note:** When using REPEAT and REPEAT  ALL commands, member numbering must be consecutive.

- \( i_1 \): Member number for which incidences are provided. Any integer number (maximum six digits) is permitted.
- \( i_2 \): Start joint number.
- \( i_3 \): End joint number.

**Note:** Use REPEAT  ALL  0 to start a set of members that will be repeated if you don't want to repeat back to the last REPEAT  ALL.

The following parameters are used for member generation only:

- \( i_4 \): Second member number to which members will be generated.
- \( i_5 \): Member number increment for generation.
- \( i_6 \): Joint number increment which will be added to the incident joints. (\( i_5 \) and \( i_6 \) will default to 1 if left out.)
- \( n \): Number of times repeat is to be carried out.
- \( m_1 \): Member number increment
- \( j_1 \): Joint number increment

The PRINT MEMBER  INFO command may be used to verify the member incidences provided or generated by REPEAT and REPEAT  ALL commands.

**Tip:** Use the Post Processing facility to verify geometry graphically.

### Example 1

MEMBER  INCIDENCES

\[
1 \ 1 \ 2 \\
2 \ 5 \ 7 \ 5 \\
7 \ 11 \ 13 \ 13 \ 2 \ 3
\]
In this example, member 1 goes from joint 1 to 2. Member 2 is connected between joints 5 and 7. Member numbers from 3 to 5 will be generated with a member number increment of 1 and a joint number increment 1 (by default). That is, member 3 goes from 6 to 8, member 4 from 7 to 9, member 5 from 8 to 10. Similarly, in the next line, member 7 will be from 11 to 13, member 9 will be from 14 to 16, 11 from 17 to 19 and 13 from 20 to 22.

Example 2

**MEMBER INCIDENCES**

<table>
<thead>
<tr>
<th>1</th>
<th>1</th>
<th>21</th>
<th>20</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>21</td>
<td>22</td>
<td>23</td>
</tr>
<tr>
<td>REPEAT 4 3 4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>21</td>
<td>25</td>
<td>39</td>
</tr>
<tr>
<td>REPEAT 3 4 4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>REPEAT ALL 9 51 20</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

This example creates the 510 members of a ten story 3 X 4-bay structure (this is a continuation of the example started in Section 5.12). The first input line creates the twenty columns of the first floor:

1 1 21 ; 2 2 22 ; 3 3 23 ; ... ; 19 19 39 ; 20 20 40

The two commands (21 21 22 23 and REPEAT 4 3 4) create 15 members which are the second floor “floor” beams running, for example, in the east-west direction:

21 21 22; 22 22 23; 23 23 24
24 25 26; 25 26 27; 26 27 28
...
33 37 38; 34 38 39; 35 39 40

The next two commands (36 21 25 39 and REPEAT 3 4 4) function similar to the previous two commands, but here create the 16 second floor “floor” beams running in the north-south direction:

36 21 25; 37 22 26; 38 23 27; 39 24 28
40 25 29; 41 26 30; 42 27 31; 43 28 32
...
48 33 37; 49 34 38; 50 35 39; 51 36 40

The preceding commands have created a single floor unit of both beams and columns, a total of 51 members. The REPEAT ALL now repeats this unit nine times, generating 459 new members and finishing the ten story structure. The member number is incremented by 51 (the number of members in a repeating unit) and the joint number is incremented by 20, (the number of joints on one floor).

**Related Links**

- [M. To add beams with new nodes](#) (on page 644)
- [G.4 Coordinate Systems and Structure Geometry](#) (on page 2295)
- [G.4.1 Global Coordinate System](#) (on page 2296)
- [G.4.2 Local Coordinate System](#) (on page 2297)
- [G.4.3 Relationship Between Global and Local Coordinates](#) (on page 2301)

**TR.13 Plate and Solid Elements**

This section describes the commands used to specify plates (i.e., shells) and solids.

**Related Links**
TR.13.1 Plate and Shell Element Incidence Specification

This set of commands is used to specify elements by defining the connectivity between joints. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD.Pro is capable of detecting multiple structures automatically.

General Format

```
ELEMENT INCIDENCES (SHELL)
i1 i2 i3 i4 (i5) ( TO i6 i7 i8 )
REPEAT n ei ji
REPEAT ALL n ei ji
```

Description

ELEMENT INCIDENCES SHELL must be provided immediately after MEMBER INCIDENCES (if any) are specified. The REPEAT command causes the previous line of input to be repeated \( n \) number of times with specified element and joint increments. The REPEAT ALL command functions similar to the REPEAT command, except that it repeats all previously specified input back to the most recent REPEAT ALL command; or to the beginning of the specification if no previous REPEAT ALL command had been issued. Use REPEAT ALL 0 0 0 to start a set of elements that will be repeated if you don’t want to repeat back to the last REPEAT ALL.

\( i1 \)  
Element number (any number up to six digits). If MEMBER INCIDENCE is provided, this number must not coincide with any MEMBER number.

\( i2 \ldots i5 \)  
Clockwise or counterclockwise joint numbers which represent the element connectivity. \( i5 \) is not needed for triangular (3 noded) elements.

The following parameters are needed if elements are to be generated:

\( i6 \)  
Last element number to which elements are generated.

\( i7 \)  
Element number increment by which elements are generated. Defaults to 1 if omitted.

\( i8 \)  
Joint number increment which will be added to incident joints. Defaults to 1 if omitted.

The following data is needed if REPEAT or REPEAT ALL commands are used to generate elements:

\( n \)  
Number of times repeat is to be carried out.

\( ei \)  
Element number increment.

\( ji \)  
Joint number increment.
The PRINT ELEMENT INFO command may be used to verify the element incidences provided or generated by REPEAT and REPEAT ALL commands.

**Tip:** Use the Post Processing facility to verify geometry graphically.

```
Example
ELEMENT INCIDENCE
1  1  2  7  6
2  3  4  8
3  8  9 11 10 TO 8
9  1  3  7 TO 14
```

**Related Links**
- *M. To draw plates connecting existing nodes* (on page 654)
- *G.5 Finite Element Information* (on page 2308)
- *G.5.1 Plate and Shell Elements* (on page 2308)
- *G.5.2 Solid Elements* (on page 2319)
- *G.5.3 Surface Elements (Deprecated)* (on page 2322)

**TR.13.2 Solid Element Incidences Specification**

Use the following commands to specify 4- through 8-noded elements, also known as solid elements. Additional information on these elements is available in *G.5.2 Solid Elements* (on page 2319).

**General Format**

The element incidences for solid elements are to be identified using the expression SOLID to distinguish them from plate/shell elements.

```
ELEMENT INCIDENCES SOLID
i1  i2  i3  i4  i5  i6  i7  i8  i9  ( TO i10 i11 i12 )
REPEAT n ei ji
REPEAT ALL n ei ji
```

**Description**

ELEMENT INCIDENCES SOLID must be provided immediately after MEMBER INCIDENCES (if any) are specified as well as after the ELEMENT INCIDENCES SHELL (if any).

- **i1** Element number
- **i2 ... i9** Joint number of the solid element
- **i10** Last element number to be generated
- **i11** Element number increment
- **i12** Joint number increment
- **n** Number of times REPEAT or REPEAT ALL is to be carried out
Element number increment

Joint number increment

Specify the four nodes of any of the faces of the solid element in a counter-clockwise direction as viewed from
the outside of the element and then go to the opposite face and specify the four nodes of that face in the same
direction used in specifying the nodes of the first face. The opposite face must be behind the first face, as defined
by the right hand rule, i.e., the opposite (back) face points to the first (front) face, which points to the viewer.

Use REPEAT ALL 0 to start a set of solids that will be repeated if you don't want to repeat back to the last
REPEAT ALL.

Example

```
ELEMENT INCIDENCES SOLID
  1  1  5  6  2 21 25 26 22 TO 3
  4 21 25 26 22 41 45 46 42 TO 6
```

Related Links
• M. To draw a solid connecting existing nodes (on page 696)
• G.5 Finite Element Information (on page 2308)
• G.5.1 Plate and Shell Elements (on page 2308)
• G.5.2 Solid Elements (on page 2319)
• G.5.3 Surface Elements (Deprecated) (on page 2322)

TR.13.3 Surface Entities Specification

Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine
will allow them but their use is not recommended.

Related Links
• G.5.3 Surface Elements (Deprecated) (on page 2322)

TR.14 Plate Element Mesh Generation

There are several methods available in STAAD to model panel type entities like walls or slabs as an assembly of
plate elements. This process is called meshing.

Two of those methods have a set of commands which can be provided in the STAAD input file.

1. Parametric mesh generator is the preferred method and is best to create this type of mesh using the
   STAAD.Pro Graphical User Interface. The aspect of this method, which enables commands to be written into
   the input file, is described in TR.14.1 Parametric Mesh Models (on page 2434).

2. The input mesh generation is described in TR.14.2 Element Mesh Generation (on page 2435) is based
   entirely on commands in the input file alone, and does not have any graphical interface for creation or
   modification.

Related Links
• G.5 Finite Element Information (on page 2308)
TR.14.1 Parametric Mesh Models

The STAAD.Pro Analytical Modeling workflow can be generate a plate element mesh using the The Parametric Models dialog (on page 2910). Openings, density lines, and density points can be included in the mesh generation.

This is the recommended method for generating meshes in STAAD.Pro as it allows you to modify the mesh parameters after creation.

Persistence of the Parametric Mesh Model Input in the STAAD.Pro Input File

When a STAAD model has portions generated from a parametric mesh model, the parametric input (which is not otherwise part of the STAAD input data) is saved within a specially designated section of the STAAD input file. This gives you the flexibility to save mesh models at any time and make modifications at a later time, such as adding an opening or a density line.

It is important that this data not be modified or removed so as to preserve the parametric model. Any changes to the portions of the model marked between the <! and !> marks may have unintended consequences to the model.

Example

Special tag based commands have been introduced to support saving of parametric mesh models as part of the STAAD input file as shown below.

```
2072 1114 1113 1160; 2073 1045 1160 1113;
ELEMENT PROPERTY
810 TO 1779 1821 TO 2073 THICKNESS 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
PARAMETRIC MODEL second_floor
MESH PARAM 0 3
MESH ORG 3 5 8
BOUNDARY 10
11 1 93 1 94 1 95 1 83 1 71 1 70 1 69 1 41 1 26 1
OPENING CIRC 72 360 96 43.2666 1
OPENING POLY 5
216 360 67.2 1 270 360 33.6 2 324 360 67.2 2 270 360 100.8 2
DENSITY POINTS 2
180 360 168 1 360 360 168 1
DENSITY LINE 0 360 168 100 180 360 168 200
DENSITY LINE 180 360 168 1 360 360 168 1
DENSITY LINE 360 360 168 1 540 360 168 1
DENSITY LINE 180 360 0 1 180 360 168 1
DENSITY LINE 180 360 168 1 180 360 336 1
DENSITY LINE 360 360 0 1 360 360 168 1
DENSITY LINE 360 360 168 1 360 360 336 1
DENSITY LINE 54 360 302.4 1 162 360 201.6 1
DENSITY LINE 216 360 201.6 1 324 360 235.2 1
```
Related Links

- [Parametric Models dialog](on page 2910)

**TR.14.2 Element Mesh Generation**

This set of commands is used to generate finite element meshes. The procedure involves the definition of super-elements, which are subsequently divided into smaller elements.

**Description**

**Tip:** It is generally recommend that you use the parametric method of mesh generation (See [TR.14.1 Parametric Mesh Models](on page 2434)) so you can modify the mesh parameters after creating the mesh.

**Note:** This method of generating element meshes is a one-type operation. That is, once you have saved the STAAD.Pro input file using these commands, the program will replace these commands with the resulting nodes and elements this command generates.

This is the second method for the generation of element incidences. If you needs to divide a big element into a number of small elements, you may use this facility which generates the joint numbers and joint coordinates, the element numbers and the element incidences automatically. Use of this feature consists of two parts:

1. **Definition of the super-element boundary points:** A super-element may be defined by either 4 boundary points or 8 boundary points (see figure below). A boundary point is denoted by a unique alphabet (A-Z in upper case or a-z in lower case) and its corresponding coordinates. Hence, any 4 or 8 of the 52 characters may be used to define the super-element boundary. If 4 points are used to define the super-element, each side of the super-element will be assumed to have a straight edge connecting the 2 points defining that side. If 8 points are used, each side will be a smooth curve connecting the 3 points defining that side.

2. **Generation of sub-elements:** define the super-element using boundary points (4 or 8 as explained above) and specify the total number of sub-elements required.
General Format

**DEFINE MESH**

\[ Ai \ x_i \ y_i \ z_i \ ( \{ CYLINDRICAL, RCYLINDRICAL \} ( x_0, y_0, z_0 ) ) \]

...

\[ Aj \ x_j \ y_j \ z_j \ ( \{ CYL, RCYL \} ( x_0, y_0, z_0 ) ) \]

**GENERATE ELEMENT { (QUADRILATERAL), TRIANGULAR }**

**MESH Ai Aj ... n1 (n2)**

**MESH Am An ... n3 (n4)**

...(up to 21 MESH input lines)

Where:

- **Ai, Aj**  Letters A-Z or letters a-z. The maximum is 52.
- **\( x_i, y_i, z_i \)**  Coordinates for boundary point \( A_i \).
- **\( x_0, y_0, z_0 \)**  Optional Cartesian coordinates of the origin for cylindrical coordinates when CYL (cylindrical) or RCYL (reverse cylindrical) options are used. This defaults to the global origin of the model (0,0,0).

The 3 fields \( x_i, y_i, z_i \) may be replaced by a joint number whose coordinates have been defined in the JOINT COORDINATE command by entering \( Ai \ JOINT \ jn \) instead.

- **\( A_i, Aj, Ak \ldots \)**  A rectangular super-element defined by four or eight boundary points.
- **\( n1 \)**  Number of elements along the side \( A_i A_j \) of the super-element. (Must not exceed 28).
- **\( n2 \)**  Number of elements along the side \( A_j A_k \) of the super-element. (Must not exceed 28).

If \( n2 \) is omitted, that is, only \( n1 \) is provided, then \( n1 \) will indicate the total number of elements within the super-element. In this case, \( n1 \) must be the square of an integer.

**Limits**

There is a limit of 21 Mesh commands. Up to 33,000 joints may be generated and up to 67,000 elements. Total number of joints in the model after this command is completed may not exceed 100,000.

**Notes**

All coordinates are in current unit system. While using this facility you has to keep the following points in mind:

1. All super-elements must be 4-noded or 8-noded. Generated elements for 4-noded super-elements will retain the straight-line edges of the super-elements, while joints of elements generated from 8-noded super-elements will lie on a curved trajectory.
A) Mesh generated for a 4-noded super-element  
B) Mesh generated for an 8-noded super-element

Figure 269: Mesh generation for super elements

2. Two super-elements, which have a common boundary, must have the same number of elements along their common boundary.

3. Sequence of super-elements - MESH commands define the super-elements. The sequence of this MESH command should be such that once one is defined, the next super-elements should be the ones connected to this. Therefore, for convenience, the first super-element should be the one, which is connected by the largest number of super-elements. In the example shown here for the tank, the bottom super-element is specified first.

4. This command must be used after the MEMBER INCIDENCE and ELEMENT INCIDENCE section and before the MEMBER PROPERTIES and ELEMENT PROPERTIES section. The elements that are created internally are numbered sequentially with an increment of one starting from the last member/element number plus one. Similarly the additional joints created internally are numbered sequentially with an increment of one starting from the last joint number plus one. It is advisable that users keep the joint numbers and member/element numbers in a sequence with an increment of one starting from one.

5. If there are members embracing a super-element which is being meshed, you must take care of the required additions/modifications in the MEMBER INCIDENCE section themselves since a few more new joints might appear on the existing common boundary as a result of meshing the super-element. See the following figure:

Figure 270: Additional joints on a super element

Note: If a member exists between points A and B, the user must breakup this member into 4 parts. Members will not be meshed automatically.

6. The sub-elements will have the same direction (Clockwise or Anti-clockwise) as the super-elements. For a super-element bounded by four points A, B, C and D, if ABCD, BCDA etc. are in clockwise direction, CBAD or DCBA etc. are in anti-clock wise direction. If the particular super-element is denoted as ABCD, all the sub-elements in it will have a clockwise element incidence in this example.

7. Element incidences of the generated sub-elements may be obtained by providing the command 'PRINT ELEMENT INFORMATION' after the 'MESH...' command in the input file.
8. If the STAAD input file contains commands for joint coordinates, member incidences, element incidences, and mesh generation, they should be specified in the following order:

```
STAAD SPACE
UNIT ...
JOINT COORDINATES
...
MEMBER INCIDENCES
...
ELEMENT INCIDENCES
...
DEFINE MESH
...
GENERATE ELEMENT
...
```

9. Newly created joints will be merged with existing joints if they are within 0.001 inches of each other.

**Example**

The following section of input illustrates the use of mesh generation facility, the user may compare this with the geometry inputs for Example Prob. No. 10:

```
STAAD SPACE TANK STRUCTURE WITH
* MESH GENERATION
UNIT . . . .
DEFINE MESH
A 0 0 0 ; B 0 20 0 ; C 20 20 0
D 20 0 0 ; E 0 0 -20 ; F 0 20 -20
G 20 20 -20 ; H 20. 0 . -20
```

*Figure 271: Mesh generation used in Example Problem 10*
**Technical Reference of STAAD Commands**

**TR.15 Redefinition of Joint and Member Numbers**

This command may be used to redefine JOINT and MEMBER numbers. Original JOINT and MEMBER numbers are substituted by new numbers.

**General Format**

```
SUBSTITUTE { { JOINT | MEMBER } { XRANGE | YRANGE | ZRANGE } | COLUMN } f1 f2 START i
```

Where:

- $f_1$ and $f_2$ are two range values of x, y, or z
- $i$ is the new starting number.
Description

Joint and member numbers can be redefined in STAAD.Pro through the use of the SUBSTITUTE command. After a new set of numbers is assigned, input and output values will be in accordance with the new numbering scheme. You can design numbering schemes that will result in simple input specification as well as easy interpretation of results. For example, all joints in first floor of a building may be renumbered as 101, 102, … etc., all second floor joints may be renumbered as 201, 202, …, etc.

Meaningful re-specification of JOINT and MEMBER numbers may significantly improve ease of interpretation of results.

This command may be in between incidence commands:

```
MEMBER INCIDENCE
...
SUBSTITUTE
ELEMENT INCIDENCE
...
```

Example

```
UNIT METER
SUBST JOINT YR 9.99 10.0 START 101
SUBST COLUMN START 901
```

Joints with Y coordinates ranging from 9.99 to 10 meters will have a new number starting from 101. Columns will be renumbered starting with the new number 901.

Related Links

- [G.22 Miscellaneous Facilities](on page 2402)

TR.16 Entities as Single Objects

In the mathematical model, beams, columns, walls, slabs, block foundations, etc. are modeled using a collection of segments, which are known by the names members, plate elements, solid elements, etc. Hence, the bottom chord of a truss may be modeled using 5 members, with each member representing the segment between points where diagonals or vertical braces meet the bottom chord.

Often, it is convenient to cluster these segments under a single name so that assignment of properties, loads, design parameters, etc. is simplified. There are presently two options in STAAD for clustering entities - Group names and Physical members.

TR.16.1 Listing of Entities by Specifying Groups

This command allows you to specify a group of entities (e.g., members, joints, elements, etc.) and save the information using a ‘group-name’. The ‘group-name’ may be subsequently used in the input file instead of a member/joint list to specify other attributes. This very useful feature allows avoiding of multiple specifications of the same member/joint list. Following is the general format required for the GROUP command.
General Format

```
START GROUP DEFINITION

followed by
(GEOMETRY)

_(group-name ) member/element/solid-list
...

or

JOINT

_(group-name ) joint-list
...

MEMBER

_(group-name ) member-list
...

ELEMENT

_(group-name ) element-list

SOLID

_(group-name ) solid-list
...

FLOOR

_(group-name ) member-list
...
```

followed by

END GROUP DEFINITION

Where:

- `group-name` = an alphanumeric name specified to identify the group. The `group-name` must start with the '_' (underscore) character and is limited to 24 characters.
- `joint-list` = the list of joints belonging to the group. TO, BY, and ALL are permitted.
- `member-list` = the list of members/joints belonging to the group TO, BY, ALL, BEAM, PLATE, and SOLID are permitted.
- `solid-list` = the list of solids belonging to the group.
- `member-list` = the list of members/joints belonging to the group TO, BY, ALL, BEAM, PLATE, and SOLID are permitted.
- ALL means all members, plates, and solids; BEAM means all beams; PLATE all plates; and SOLID all solids.

Notes

1. The GROUP definition must start with the `START GROUP DEFINITION` command and end with the `END` command.
2. More than one GROUP name may be specified within the same definition specification.
3. The words JOINT, MEMBER, ELEMENT, FLOOR, or SOLID may be provided if the you wish to identify the group name and lists with those specific items. However, if the group name and list is merely a means of grouping together more than one type of structural component under a single heading, the word GEOMETRY may be provided. In the absence of any of those words (GEOMETRY, JOINT, MEMBER, ELEMENT, FLOOR, or SOLID), the list is assumed to be that for GEOMETRY.

4. The same joint or member/element number may be included in up to four groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.

5. If two or more consecutively entered groups have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.

6. A member group may be used in lieu of a member-list with virtually any command which requires member lists, such as MEMBER LOADS, steel and concrete parameters, etc. There is one place however where a MEMBER GROUP will not suffice, and that is for defining panels during a FLOOR LOAD assignment.

In Applying FLOOR LOAD onto a Floor Group (on page 2674), a panel has to be specified using a FLOOR GROUP, not a MEMBER GROUP. A FLOOR GROUP is not accepted in lieu of a member-list for any other command.

7. The maximum number of group allowed in an input file is equal to the total number of member, plates, solid, and nodes times the number of duplicate entities allowed to be in different group, which is by default 10 (may be changed using the SET GROUP DUPLICATE i command as described in TR.5 Set Command Specification (on page 2413)).

For example, if a model has 10 members, 3 plates and 2 solids, and 100 nodes. The maximum number of groups could be $10 \cdot (10 + 3 + 2 + 100) = 1150$.

**Example 1**

```
START GROUP DEFINITION
  _TRUSS 1 TO 20 25 35
  _BEAM 40 TO 50
END
MEMBER PROPERTIES
  _TRUSS TA LD L40304
  _BEAM TA ST W12X26
```

**Example 2**

```
START GROUP DEFINITION
  JOINT
  _TAGA 1 TO 10
  MEMBER
  _TAGB 40 TO 50
  GEOMETRY
  _TAGC 101 TO 135
END
MEMBER PROPERTIES
  _TAGB TA LD L40304
  _TAGC TA ST W12X26
```

**Related Links**

- *M. To create a group from a selection* (on page 704)
- *G.4 Coordinate Systems and Structure Geometry* (on page 2295)
STAAD.Pro allows grouping analytical predefined members into physical members using a special member group `PMEMBER`. This command defines a group of analytical, collinear members with same cross section and material property.

To model using `PMEMBER`, you need to model regular analytical members and then, group those together.

While creating a `PMEMBER`, the following pre-requisites apply:

1. Existence of the analytical members in the member-list.
2. Selected members should be interconnected.
3. The selected individual members must be collinear (adjacent analytical members must lie within 5°).
4. Local axis of the individual members comprising the physical member should be identical (i.e., x, y and z are respectively parallel and in same sense).

   **Tip:** You may select the **Beam Tools > Beam Incidence** tool in the **Geometry Tools** group on the **Utilities** ribbon tab in the STAAD.Pro Analytical Modeling workflow for modifying analytical members which are pointing in the wrong direction.

5. A member in one physical member group can not be part of any other physical member group.

   **Note:** The `PMEMBER` option can only be used for codes which explicitly support that command. Refer to the appropriate design code specification to confirm if that code supports `PMEMBER` design.

### Description

A `PMEMBER` can be created either in the Analytical Modeling workflow or in the Steel Designer workflow. Analytical Modeling workflow and Steel Designer workflow `PMEMBERs` will be labeled as M and D, respectively. Modeling workflow `PMEMBERs` allow variable cross-sections. Steel Designer workflow allows importing of `PMEMBERs` created in the modeling mode.

To define a Physical Member, the following command is used after the `MEMBER INCIDENCE` command:

```
DEFINE PMEMBER
```

**Example**

```
JOINT COORDINATE
1 0 0 6 10.0 0 0
MEMBER INCIDENCE
1 1 2 5
DEFINE PMEMBER
1 TO 5 PMEMB 1
```
To define the member property of a Physical Member, the following command is used:

\texttt{PMEMBER PROPERTY pmember-list PRIS ...}

The physical member supports all types of member properties available in STAAD.Pro. If multiple definitions of member properties for a particular analytical member is encountered (e.g., analytical member properties is defined twice, once via \texttt{PMEMBER PROP} command and again via the \texttt{MEMBER PROP} command, then the \texttt{MEMBER PROP} command will override the \texttt{PMEMBER PROP} definition.

To define the material constants of a physical member, the following command is used:

\texttt{PMEMBER CONSTANTS E CONCRETE pmember-list}

\texttt{DEN CONCRETE pmember-list}

... Any member, which is a part of any \texttt{PMEMBER} is not allowed to be assigned constants explicitly.

\textbf{Note:} Loads are applied directly to physical members using the \texttt{PMEMBER LOAD} command. Refer to \texttt{TR.32.2.1 PMember Load Specification} (on page 2655) for details.

Design parameters are available for use with PMEMBERS by using the PMEB list. The following syntax is used:

\texttt{parameter value PMEMB pmember-list}

\textbf{Example}

\begin{verbatim}
RATIO 1.05 PMEMP 1 2
\end{verbatim}

\textbf{Note:} There is not option to specify \texttt{ALL} for a PMEB list, except for \texttt{CODE CHECK} or \texttt{SELECT} commands.

After the analysis, the Post Analysis results of a PMEMBER can be seen by using the following command:

\texttt{PRINT PMEMBER FORCE}

This command will produce member forces for all the analytical members in the group.

\textbf{Related Links}

- \textit{M. Physical Members} (on page 649)

\section*{TR.17 Rotation of Structure Geometry}

This command may be used to rotate the currently defined joint coordinates (and the attached members/elements) about the global axes. The rotated configuration is used for analysis and design. While specifying this command, the sense of the rotation should conform to the right hand rule.

\textbf{General Format}

\texttt{PERFORM ROTATION *{ X d1 | Y d2 | Z d3 }}
Where:

\( d_1, d_2, d_3 \) are the rotations (in degrees) about the X, Y, and Z global axes, respectively.

Example

PERFORM ROTATION X 20 Z -15

Related Links

- *G.22 Miscellaneous Facilities* (on page 2402)
- *G.4 Coordinate Systems and Structure Geometry* (on page 2295)
- *G.4.1 Global Coordinate System* (on page 2296)
- *G.4.2 Local Coordinate System* (on page 2297)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2301)

TR.18 Inactive/Delete Specification

This set of commands may be used to temporarily inactivate or permanently delete specified JOINTs or MEMBERs.

General Format

```
INACTIVE { MEMBERS member-list | ELEMENTS element-list }
DELETE { MEMBERS member-list | JOINTS joint-list }
```

Description

These commands can be used to specify that certain joints or members be deactivated or completely deleted from a structure. The INACTIVE command makes the members and elements temporarily inactive; the user must re-activate them during the later part of the input for further processing. The DELETE command will completely delete the members/elements from the structure; you cannot re-activate them. The Delete Joint command must be immediately after the Joint Coordinates. The DELETE commands must be provided immediately after all member/element incidences are provided and before any INACTIVE commands.

Notes

a. The DELETE MEMBER command will automatically delete all joints associated with deleted members, provided the joints are not connected by any other active members or elements.

b. This command will also delete all the joints, which were not connected to the structure in the first place. For example, such joints may have been generated for ease of input of joint coordinates and were intended to be deleted. Hence, if a DELETE MEMBER command is used, a DELETE JOINT command should not be used.

c. The DELETE MEMBER command is applicable for deletion of members as well as elements. If the list of members to be deleted extends beyond one line, it should be continued on to the next line by providing a blank space followed by a hyphen (-) at the end of the current line.
d. If inactivating members causes joints to become unconnected in space (i.e., orphaned), a warning stating that those joints are not connected by any members or elements will be displayed in the output file. These warnings can normally be ignored.

e. The inactivated members may be restored for further processes (such as an analysis or design for a 2nd set of load cases) by using the CHANGE command. See TR.37 Analysis Specification (on page 2795) and Example EX. US-4 Inactive Members in a Braced Frame (on page 4390) for more information.

f. The DELETE MEMBER command should be used to delete elements too. Specify the command as DELETE MEMBER \( j \), where \( j \) is the element number of the element you wish to delete. In the example shown below, 29 to 34 and 43 are element numbers.

g. Loads that have been defined on members declared as INACTIVE members will not be considered in the analysis. This applies to SELFWEIGHT, MEMBER LOADS, PRESTRESS, and POSTSTRESS LOADS, TEMPERATURE LOADS, etc.

For those joints that become orphaned (see note d above), loads applied to such joints, either explicitly through the JOINT LOAD command, or generated by some means as in moving, wind or seismic load generation, will be "lost", meaning, they will not get considered for the analysis for those load cases for which the associated member is inactivated.

h. The DELETE JOINT command must be specified before all incidence commands such as MEMBER INCIDENCE, ELEMENT INCIDENCE, etc.

Example

<table>
<thead>
<tr>
<th>INACTIVE MEMBERS 5 7 TO 10</th>
</tr>
</thead>
<tbody>
<tr>
<td>DELETE MEMBERS 29 TO 34 43</td>
</tr>
</tbody>
</table>

Related Links

- G.19 Multiple Analyses (on page 2401)
- EX. US-4 Inactive Members in a Braced Frame (on page 4390)
- EX. UK-4 Inactive Members in a Braced Frame (on page 4670)

TR.19 User Steel Table Specification

STAAD.Pro allows you to create and use customized steel section tables for use in the property specification, code checking, and member selection. This set of commands may be used to create the tables and provide necessary data.

General Format

<table>
<thead>
<tr>
<th>START USER TABLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>TABLE 11 (fn )</td>
</tr>
<tr>
<td>section-type</td>
</tr>
</tbody>
</table>

The following commands are repeated for each section within the user-table.

<table>
<thead>
<tr>
<th>section-name</th>
</tr>
</thead>
<tbody>
<tr>
<td>property-spec</td>
</tr>
</tbody>
</table>
For General sections, the following may be specified for each section:

(\textbf{PROFILE\_POINTS})

\[zp1 \ yp1 \ zp2 \ yp2 \ \ldots \ zpn \ ypn\]

(\textbf{STRESS\_LOCATIONS})

\[zs1 \ ys1 \ zs2 \ ys2 \ zs3 \ ys3 \ zs4 \ ys4\]

All tables end with the following command:

\textbf{END}

Where:

\textbf{i1} \quad \text{table number (1 to 99). During the analysis process, the data in each user provided table is stored in a corresponding file with an extension .U0n. For example, the data of the 5th table is stored in .U05. The first part of the input file name is the same as that of the STAAD input file. These files are located in the same working directory as the input file. Hence, they may later be used as external user provided tables for other input files.}

\textbf{fn} \quad \text{external file path and file name containing the section name and corresponding properties (up to 72 characters). If an external file is used, the no other data should be provided for this table.}

\textbf{section-type} \quad \text{a steel section name including: WIDE\_FLANGE, CHANNEL, ANGLE, DOUBLE\_ANGLE, TEE, PIPE, TUBE, GENERAL, ISECTION, & PRISMATIC.}

\textbf{section-name} \quad \text{Any user designated section name, use 1 to 36 characters observing the following rules:}

- Only alphanumeric characters and digits are allowed for defining section names (i.e., spaces, asterisks, question marks, colons, underscores, semi-colons, etc. are not permitted).
- The first three characters of the section names in pipe and tube tables must be PIP and TUB, respectively.
- The first the characters of the section names in all other tables cannot start with PIP or TUB.
- A name can be duplicated in different tables, but must be unique within any individual table.

\textbf{property-spec} \quad \text{Properties for the section. The requirements are different for each section type as follows. Shear areas }AY\text{ and }AZ\text{ must be provided to ensure proper shear stress or shear strength calculations during design.}

The default length units for properties are the current units. If a UNIT command is entered within the User Table in the input file then those units become the current units. However, a UNIT command on an external file only affects that file and has no effect on the units in subsequent input file commands. You may specify the desired length unit by using the UNIT command as the first command in the table (see example following this description).

If data is from input file, then use up to three lines of input per \textbf{property-spec} (end all but last with a hyphen, -).

\section*{General Section Definitions}

The commands \textbf{PROFILE\_POINTS} and \textbf{STRESS\_LOCATIONS} can be used with a user-table general shape only. These define the vertices of a closed shape and the four locations to calculate stress, respectively.
are the X and Y pair of local coordinates which describe the points of the section. The points should be provided in either clockwise or counter-clockwise order.

are the X and Y pair of local coordinates which indicate where the stress is calculated. It is recommended that these start with the top, left-most point. The extreme points of the cross-section are then specified in a clockwise order around the section. Though any coordinates may be entered, points should lie on the section defined in order to represent true stress values.

Note: Hyphens (i.e., dashes) may not be used to break a line of PROFILE_POINTS or STRESS_LOCATIONS data.

Caution: Do not use comments (i.e., lines that start with an asterisk) within the PROFILE_POINTS and STRESS_LOCATIONS definitions, as these lines will not be ignored by the STAAD engine. This will result in unintended sections or errors.

Tip: Using the Graphical Interface is recommended for entering these points, as they may be entered with respect to any local coordinate system and will be translated to the section's center of mass and the section property-spec values will be calculated automatically. See M. To create a general section (on page 735) for details.

Example

START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
P24X55-abcdefghijklmnopqrstuvwxyz111
16.2 23.57 0.375 7.005 0.505 1350 29.1 1.00688 8.83875 7.07505
P24X56
18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
END
START USER TABLE
TABLE 2
GENERAL
L6x6x1
11 6 0 6 0 35.4621 35.4621 3.40525 8.57326 8.57326 5.09239 -
5.01615 21.4489 10.9903 8.98521 6
PROFILE_POINTS
-1.86364 4.13636 -0.863636 4.13636 -0.863636 -0.863636 4.13636 -0.863636 -0.863636
4.13636 -1.86364 -1.86364 -1.86364
STRESS_LOCATIONS
-1.8636 4.13636 -0.8638 -0.8636 4.13636 -1.8636 -1.8636 -1.8636
END
... member PROPERTY
27 UPTABLE 1 P24X55-abcdefghijklmnopqrstuvwxyz111
39 UPTABLE 1 P24X56

The following example uses an external file, C:\Structural Models\My_Profile.upt (exported from Section Wizard), as the data source for a general section:

START USER TABLE
TABLE 1 C:\STRUCTURAL MODELS\MY_PROFILE.upt

Related Links
• G.6.3 User-Provided Steel Table (on page 2325)
TR.19.1 Wide Flange

General Format

For steel wide flange shapes:

\[
property-spec = AX \ D \ TW \ WF \ TF \ IZ \ IY \ IX \ AY \ AZ (WF1) (TF1)
\]

For composite wide flange shapes:

\[
property-spec = -AX \ D \ TW \ WF \ TF \ IZ \ IY \ IX \ AY \ AZ (WF1) (TF1)
\]

For composite wide flange shapes with bottom cover plate:

\[
property-spec = -CFR \ CFL \ CFT \ MR
\]

\[
property-spec = -CFR \ CFL \ CFT \ MR
\]

\[
BPR \ BPL \ BPT
\]

**Note:** There is a leading dash on the first line of section property specifications when composite slab properties are specified and again at the beginning of the composite slab properties when bottom cover plate dimensions are specified.

- **AX**  Cross section area
- **D**  Depth of the section
- **TW**  Thickness of web
- **WF**  Width of the top flange (or both flanges when WF1 is not specified)
- **TF**  Thickness of top flange (or both flanges when WF1 is not specified)
- **IZ**  Moment of inertia about local z-axis (usually strong axis)
IY  Moment of inertia about local y-axis
IX  Torsional constant
AY  Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
AZ  Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.
WF1 (Optional) Width of the bottom flange. The width of the top flange will be used if this value is not specified.
TF1 (Optional) Thickness of bottom flange. The thickness of the top flange will be used if this value is not specified.

The following option parameters are used to include a composite concrete slab. If included, these must be on a separate line following a dash, -, on the line containing the required section parameters.

CFR  Width of the composite slab to the right of the web center line
CFL  Width of the composite slab to the left of the web center line
CFT  Thickness of the composite slab
MR  Modular ratio of the concrete in the composite slab

The following option parameters are used to include a bottom flange cover plate.

Tip: Bottom cover plates can only be added in association with a composite slab.

BPR  Width of the additional bottom flange plate to the left of the web center line
BPL  Width of the additional bottom flange plate to the right of the web center line
BPT  Thickness of the additional bottom flange plate

---

Example

```
START USER TABLE
TABLE 1
UNIT MMS
WIDE FLANGE
UNEQUAL_FLANGE_I
16855 600 10 405 15 1.10087e+009 1.21335e+008 1.13626e+006 6000 7450 300 17
UNEQUAL_FLANGE_COMP_I
-16855 600 10 405 15 1.10087e+009 1.21335e+008 1.13626e+006 6000 7450 - 300 17
250 350 75 9.1
UNEQUAL_FLANGE_COMP_BOTPLT_I
-16855 600 10 405 15 1.10087e+009 1.21335e+008 1.13626e+006 6000 7450 - 300 17
-250 350 75 9.1
120 100 25
END
```
TR.19.2 Channel

*Figure 274: Channel section*

property-spec = AX D TW WF TF IZ IY IX CZ AY AZ

Where:

- **AX** Cross section area
- **D** Depth of the section
- **TW** Thickness of web
- **WF** Width of flange
- **TF** Thickness of flange
- **IZ** Moment of inertia about local z-axis (usually strong axis)
- **IY** Moment of inertia about local y-axis
- **IX** Torsional constant
- **CZ** Distance from back of web to center of gravity (C.G.) of the shape along the local z-axis.
- **AY** Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
- **AZ** Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.
TR.19.3 Angle

**Figure 275: Angle section**

<table>
<thead>
<tr>
<th><em>property-spec</em></th>
<th>D</th>
<th>WF</th>
<th>TF</th>
<th>R</th>
<th>AY</th>
<th>AZ</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>D</strong></td>
<td>Depth of angle (i.e., length of the leg along the local y-axis)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>WF</strong></td>
<td>Width of angle (i.e., length of the leg along the local z-axis)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>TF</strong></td>
<td>Thickness of angle leg</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>R</strong></td>
<td>Radius of gyration about principal axis, shown as r(Z-Z) in the AISC manual (this must not be zero)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>AY</strong></td>
<td>Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>AZ</strong></td>
<td>Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Tip:** If adding an user-provided table angle in the user interface, you can have the program calculate R, AY, and AZ values.

TR.19.4 Double Angle

**Figure 276: Double angle section**

<table>
<thead>
<tr>
<th><em>property-spec</em></th>
<th>D</th>
<th>WF</th>
<th>TF</th>
<th>SP</th>
<th>IZ</th>
<th>IY</th>
<th>IX</th>
<th>CY</th>
<th>AY</th>
<th>AZ</th>
<th>RVV</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>D</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>WF</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>TF</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>SP</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>IZ</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>IY</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>IX</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>CY</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>AY</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>AZ</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>RVV</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Technological Reference of STAAD Commands**

**TR.19 User Steel Table Specification**

- **D** Depth of angle (i.e., length of the leg along the local y-axis)
- **WF** Width of angle (i.e., length of the leg along the local z-axis)
- **TF** Thickness of angle leg
- **SP** Space between angles.
- **IZ** Moment of inertia about local z-axis (usually strong axis)
- **IY** Moment of inertia about local y-axis
- **IX** Torsional constant
- **CY** Distance from the face of angles to the center of gravity (C.G.) along the local y-axis.
- **AY** Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
- **AZ** Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.
- **RVV** The radius of gyration about the minor principal axis for single angles - z-z axis for TA ST angles, and y-y axis for TA RA angles.

### TR.19.5 Tee

**Figure 277: Tee section**

<table>
<thead>
<tr>
<th>property-spec</th>
<th>AX</th>
<th>D</th>
<th>WF</th>
<th>TF</th>
<th>TW</th>
<th>IZ</th>
<th>IY</th>
<th>IX</th>
<th>CY</th>
<th>AY</th>
<th>AZ</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>AX</strong></td>
<td>Cross section area</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>D</strong></td>
<td>Depth of the section</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>WF</strong></td>
<td>Width of flange</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>TF</strong></td>
<td>Thickness of flange</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>TW</strong></td>
<td>Thickness of web</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>IZ</strong></td>
<td>Moment of inertia about local z-axis (usually strong axis)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>IY</strong></td>
<td>Moment of inertia about local y-axis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>IX</strong></td>
<td>Torsional constant</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*STAAD.Pro 2453 User Manual*
**Technical Reference of STAAD Commands**

**TR.19 User Steel Table Specification**

- **CY**  Distance from back of web to center of gravity (C.G.) of the shape along the local z-axis.
- **AY**  Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
- **AZ**  Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

**TR.19.6 Pipe**

![Pipe Diagram]

**property-spec = OD ID AY AZ**

- **OD**  Outer diameter
- **ID**  Inner diameter
- **AY**  Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
- **AZ**  Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

**TR.19.7 Tube**

![Tube Diagram]

**property-spec = AX D WF TF IZ IY IX AY AZ**

- **AX**  Cross section area
- **D**  Depth of the section
- **WF**  Section width
### TR.19.8 General

The following cross-sectional properties should be used for this section-type. This facility allows you to specify a built-up or unconventional steel section. Provide both the Y and Z parameters for design or code checking.

**Note:** STAAD.Pro can perform some code checks on a General section where as a Prismatic section (TR.20.2 Prismatic Property Specification (on page 2465)) is for analysis only.

#### property-spec

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AX</td>
<td>Cross section area</td>
</tr>
<tr>
<td>D</td>
<td>Depth of the section</td>
</tr>
<tr>
<td>TD</td>
<td>Thickness associated with section element parallel to depth (usually web). To be used to check depth/thickness ratio</td>
</tr>
<tr>
<td>B</td>
<td>Width of the section</td>
</tr>
<tr>
<td>TB</td>
<td>Thickness associated with section element parallel to flange. To be used to check width/thickness ratio</td>
</tr>
<tr>
<td>IZ</td>
<td>Moment of inertia about local z-axis (usually strong axis)</td>
</tr>
<tr>
<td>IY</td>
<td>Moment of inertia about local y-axis</td>
</tr>
<tr>
<td>IX</td>
<td>Torsional constant</td>
</tr>
<tr>
<td>SZ</td>
<td>Section modulus about local z-axis</td>
</tr>
<tr>
<td>SY</td>
<td>Section modulus about local y-axis</td>
</tr>
<tr>
<td>AY</td>
<td>Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.</td>
</tr>
<tr>
<td>AZ</td>
<td>Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.</td>
</tr>
<tr>
<td>PZ</td>
<td>Plastic modulus about local z-axis</td>
</tr>
<tr>
<td>PY</td>
<td>Plastic modulus about local y-axis</td>
</tr>
</tbody>
</table>

---

**Technical Reference of STAAD Commands**

TR.19 User Steel Table Specification
**HSS**  Warping constant for lateral torsional buckling calculations

**DEE**  Depth of web. For rolled sections, distance between fillets should be provided

**Note:** Properties PZ, PY, HSS, and DEE must be provided for code checking or member selection per plastic and limit state based codes (AISC LRFD, British, French, German and Scandinavian codes). For codes based on allowable stress design (AISC-ASD, AASHTO, Indian codes), zero values may be provided for these properties.

You can also specify points to define a general section. Refer to [TR.19 User Steel Table Specification](on page 2446) for details. When this method is used, the section property-spec values can be calculated automatically by the graphical interface.

**TR.19.9 I Section**

This section type may be used to specify a generalized I-shaped section. The cross-sectional properties required are listed below. This section can be used to specify tapered I-shapes.

**ISECTION**

- **section-name**
- **DWW**
- **TWW**
- **DW1**
- **BFF**
- **TFF**
- **TF1**
- **AYF**
- **AZF**
- **XIF**

**CFL**

**CFR**

**CFT**

**DWW**

**TWW**

**Section A-A**

**Elevation**

*Figure 279: I section and tapered web*

- **DW**  Depth of section at start node
- **TWW**  Thickness of web
The following option parameters are used to include a composite concrete slab. If included, these must be on a separate line following a dash, -, on the line containing the required section parameters.

- **CFR** Width of the composite slab to the right of the web center line
- **CFL** Width of the composite slab to the left of the web center line
- **CFT** Thickness of the composite slab
- **MR** Modular ratio of the concrete in the composite slab

### Notes

**DWW** should never be less than **DWW1**. Therefore, you must provide the member incidences accordingly.

You are allowed the following options for the values **AYF**, **AZF**, and **XIF**.

- If positive values are provided, they are used directly by the program.
- If zero is provided, the program calculates the properties using the following formula.
  - **AYF** = \( D \times T_{WW} \)
  - **AZF** = 0.66 \( (BFF \times TFF) + (BFF_1 \times TFF_1) \)
  - **XIF** = \( \frac{1}{3} \) \( (BFF \times TFF^3) + (DEE \times T_{WW}^3) + (BFF_1 \times TFF_1^3) \)

**DEE** = Depth of web of section

If negative values are provided, they are applied as factors on the corresponding value(s) calculated by the program using the above formula. The factor applied is always the absolute of the value provided. For example, if you specify a value of **XIF** as -1.3, then the program will multiply the value of **XIF**, calculated by the above formula, by a factor of 1.3.

### TR.19.10 Prismatic

The property-spec for the **PRISMATIC** section-type is as follows:

- **AX** Cross-section area
- **IZ** Moment of inertia about the local z-axis
**Technical Reference of STAAD Commands**

**TR.19 User Steel Table Specification**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>IY</strong></td>
<td>Moment of inertia about the local y-axis</td>
</tr>
<tr>
<td><strong>IX</strong></td>
<td>Torsional constant</td>
</tr>
<tr>
<td><strong>AY</strong></td>
<td>Shear area for shear parallel to local y-axis.</td>
</tr>
<tr>
<td><strong>AZ</strong></td>
<td>Shear area for shear parallel to local z-axis.</td>
</tr>
<tr>
<td><strong>YD</strong></td>
<td>Depth of the section in the direction of the local y-axis.</td>
</tr>
<tr>
<td><strong>ZD</strong></td>
<td>Depth of the section in the direction of the local z-axis</td>
</tr>
</tbody>
</table>

**Note:** When listing multiple shapes, those section names must be provided in ascending order by weight since the member-selection process uses these tables and the iteration starts from the top.

**Example**

```
START USER TABLE
TABLE 1
UNIT . . .
WIDE FLANGE
W14X30
  8.85  13.84  .27  6.73  .385  291.  19.6  .38  0  0
W21X50
  14.7  20.83  .38  6.53  .535  984  24.9  1.14  7.92 0
W14X109
  32.  14.32  .525  14.605  .86  1240  447  7.12  7.52  0
TABLE 2
UNIT . . .
ANGLES
L25255
  2.5  2.5  .3125 .489  0  0
L40404
  4.  4.  .25  .795  0  0
END
```

**TR.19.11 Using Reference Table Files**

The User-Provided Steel Tables may be created and maintained as separate files. The same files may be used for all models using sections from these tables. These files should reside in the same directory where the input file is located.

```
START USER TABLE
TABLE 1 TFILE1
TABLE 2 TFILE2
END
```

Where *TFILE1* and *TFILE2* are names of files which must be created prior to running STAAD, and where the file *TFILE1* contains the following:

On each file the first table should contain a **UNIT** command.

```
UNIT . . .
WIDE FLANGE
```
TR.20 Member Property Specification

This set of commands may be used for specification of section properties for frame members.

The options for assigning properties come under two broad categories:

1. Those which are specified from built-in property tables supplied with the program, such as for steel, aluminum and timber.
2. Those which are not assigned from built-in tables, but instead are specified on a project-specific basis, such as for concrete beams and columns, or custom-made sections for industrial structures.

Properties which are specified from built-in property tables

1. General format for standard steel (hot rolled):

   MEMBER PROPERTIES { AMERICAN | APLAPOLLOTUBE | AUSTRALIAN | BRAZILIAN | BRITISH | CANADIAN | CHINESE | DUTCH | EUROPEAN | FRENCH | GERMAN | INDIAN | JAPANESE | JINDAL | KOREAN | MEXICAN | RUSSIAN | SAFRICAN | SPANISH | STOASCHM | TATASTRUCTURA | VENEZUELAN }

   member-list { TABLE type-spec section-name-in-table (additional-spec) | ASSIGN profile-spec }

   AMERICAN, BRITISH, EUROPEAN (etc.) option will instruct the program to read properties from the appropriate steel table. The default depends on the country of distribution.

   - Refer to TR.20.1 Assigning Properties from Steel Tables (on page 2461) for type-specs and additional-specs.
   - Refer to TR.20.5 Assign Profile Specification (on page 2470) for ASSIGN profile-spec.
   - contain information on the section types which can be assigned for the various countries named in the above list.
   - Refer to TR.20.6 Examples of Member Property Specification (on page 2472) for examples.

   The MEMBER PROPERTY command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).
2. General format for cold formed steel:

```plaintext
MEMBER PROPERTIES { BUTLER | COLD AMERICAN | COLD AUSTRALIAN | COLD BRITISH | COLD EUROPEAN | COLD INDIAN | COLD JAPANESE | COLD RUSSIAN | KINGSPAN | LYSAGHT | RCECO }
```

`member-list TABLE ST section-name-in-table`

Refer to D. Design Codes (on page 1366) information on the section types which can be assigned for the various countries/organizations named in the above list.

3. General format for steel joist:

```plaintext
MEMBER PROPERTIES SJIJOIST
```

`member-list TABLE ST section-name-in-table`

G.6.6 Steel Joist and Joist Girders (on page 2326) contains information on the joist types which can be assigned from the Steel Joist Institute's tables.

4. General format for Aluminum:

```plaintext
MEMBER PROPERTIES ALUMINUM
```

`member-list TABLE ST section-name-in-table`

D1.H. American Codes - Aluminum Design per 1994 ADM (on page 1543) contains information on the section types which can be assigned for the aluminum table in the above list.

5. General format for Timber:

```plaintext
MEMBER PROPERTIES { AITC | TIMBER CANADIAN }
```

`member-list TABLE ST section-name-in-table`

D1.H. American Codes - Aluminum Design per 1994 ADM (on page 1543) contains information on the section types which can be assigned for the above list.

Properties that are not specified from built-in property tables

```plaintext
MEMBER PROPERTIES
```

`member-list { PRISMATIC property-spec | TAPERED argument-list | UPTABLE i, section-name }
```

- Refer to TR.20.2 Prismatic Property Specification (on page 2465) for specification of PRISMATIC properties.
- Refer to TR.20.3 Tapered Member Specification (on page 2468) for specification of TAPERED members.
- Refer to TR.20.4 Property Specification from User Provided Table (on page 2469) for specification from UPTABLES.
- Refer to TR.20.6 Examples of Member Property Specification (on page 2472) for examples.

The MEMBER PROPERTY command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).

**Related Links**

TR.31.2.14 IBC 2015 Seismic Load Definition

- TR.31.2.14 IBC 2015 Seismic Load Definition (on page 2600)
- G.6.2 Built-In Steel Section Libraries (on page 2325)
- M. To add a new table section property (on page 730)
- G.6 Member Properties (on page 2322)
TR.20.1 Assigning Properties from Steel Tables

The following commands are used for specifying section properties from built-in steel table(s). The section type specification is followed by additional specifications as needed.

General Format

\[
\text{type-spec . table-name additional-spec}
\]

Where:

\[
\text{type-spec} = \{ \text{ST | RA | D | LD | SD | T | CM | TC | BC | TB | FR} \}
\]

Table 231: Type-specs for various profile types

<table>
<thead>
<tr>
<th>type-spec</th>
<th>Description</th>
<th>Diagram</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST</td>
<td>single section from the standard built-in tables</td>
<td></td>
</tr>
<tr>
<td>RA</td>
<td>single angle with reverse Y-Z axes (refer to G.4.2 Local Coordinate System (on page 2297))</td>
<td></td>
</tr>
<tr>
<td>D</td>
<td>double profile. In the case of channels, back-to-back</td>
<td></td>
</tr>
<tr>
<td></td>
<td>The spacing between shapes is provided using the SP additional specification.</td>
<td></td>
</tr>
<tr>
<td>type-spec</td>
<td>Description</td>
<td>Diagram</td>
</tr>
<tr>
<td>----------</td>
<td>--------------------------------------------------</td>
<td>---------</td>
</tr>
<tr>
<td>LD</td>
<td>double angle with long legs back-to-back</td>
<td><img src="ld.png" alt="Diagram" /></td>
</tr>
<tr>
<td>SD</td>
<td>double angle with short legs back-to-back</td>
<td><img src="sd.png" alt="Diagram" /></td>
</tr>
<tr>
<td>T</td>
<td>tee section cut from I shaped section</td>
<td><img src="t.png" alt="Diagram" /></td>
</tr>
<tr>
<td>CM</td>
<td>composite section, available for I shaped sections</td>
<td><img src="cm.png" alt="Diagram" /></td>
</tr>
<tr>
<td>TC</td>
<td>section with top cover plate</td>
<td><img src="tc.png" alt="Diagram" /></td>
</tr>
<tr>
<td>BC</td>
<td>section with bottom cover plate</td>
<td><img src="bc.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>
### Technical Reference of STAAD Commands

#### TR.20 Member Property Specification

<table>
<thead>
<tr>
<th>type-spec</th>
<th>Description</th>
<th>Diagram</th>
</tr>
</thead>
<tbody>
<tr>
<td>TB</td>
<td>section with both top and bottom cover plate top plate dimensions are described using WP and TH parameter values and bottom plate dimensions are described using BW and BT parameter values</td>
<td><img src="image1.png" alt="Diagram" /></td>
</tr>
<tr>
<td>FR</td>
<td>front-to-front (i.e., toe-to-toe) channels. <strong>Note:</strong> Spacing between the channels must be provided using the SP option mentioned in the additional spec specification described below.</td>
<td><img src="image2.png" alt="Diagram" /></td>
</tr>
<tr>
<td>SA</td>
<td>double angle in a star arrangement (heel to heel)</td>
<td><img src="image3.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

### Table 232: Additional specifications for steel sections

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SP f1</td>
<td>Either:</td>
</tr>
<tr>
<td></td>
<td>• Spacing between double angles, double channels, or double wide-flange sections (i.e., D, LD, SD, or FR). The default spacing is 0.0.</td>
</tr>
<tr>
<td></td>
<td>• Rib height for composite sections</td>
</tr>
</tbody>
</table>

---

*Table section name like W8X18, C15X33 etc.*

The documentation on steel design per individual country codes contains information regarding their steel section specification also. For details on specifying sections from the American steel tables, refer to [D1.A.4 Built-in Steel Section Library](on page 1368).
### Variable Description

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WP (f_2)</td>
<td>Width of top cover plate for I shaped sections with cover plate.</td>
</tr>
<tr>
<td>TH (f_3)</td>
<td>Thickness of plates or tubes (top plate for I shaped sections)</td>
</tr>
<tr>
<td>WT (f_4)</td>
<td>Width of tube.</td>
</tr>
<tr>
<td>DT (f_5)</td>
<td>Depth of tube.</td>
</tr>
<tr>
<td>OD (f_6)</td>
<td>Outer diameter of pipe.</td>
</tr>
<tr>
<td>ID (f_7)</td>
<td>Inside diameter of pipe.</td>
</tr>
<tr>
<td>CT (f_8)</td>
<td>Concrete thickness of the concrete for composite sections.</td>
</tr>
<tr>
<td>FC (f_9)</td>
<td>Compressive strength of concrete for composite sections.</td>
</tr>
<tr>
<td>CW (f_{10})</td>
<td>Concrete width for composite sections.</td>
</tr>
<tr>
<td>CD (f_{11})</td>
<td>Concrete density for composite sections (default is 150 pounds per cubic ft.)</td>
</tr>
<tr>
<td>BW (f_{12})</td>
<td>With of bottom cover plate for I shaped sections with cover</td>
</tr>
<tr>
<td>BT (f_{13})</td>
<td>Thickness of cover plates for I shaped sections with cover</td>
</tr>
</tbody>
</table>

Refer to [G.6.2 Built-In Steel Section Libraries](on page 2325) for more information.

### Examples

```
UNIT ...
MEMBER PROPERTIES
1 TO  5  TABLE  ST  W8X31
  9 10  TABLE  LD  L40304  SP  0.25
12 TO 15  PRISMATIC  AX  10.0  IZ  1520.0
17 18 TA  ST  PIPE  OD  2.5  ID  1.75
20 TO 25 TA  ST  TUBE  DT  12.  WT  8.  TH  0.5
27 29 32 TO 40 42  PR  AX  5.  IZ  400.  IY  33. -
  IX  0.2  YD  9.  ZD  3.
43 TO 47 UPT  1  W10X49
50 51 UPT  2  L40404
52 TO 55  ASSIGN  COLUMN
56 TA  TC  W12X26  WP  4.0  TH  0.3
57 TA  CM  W14X34  CT  5.0  FC  3.0
```
Notes

All values f1 - f9 must be supplied in current units.

Some important points to note in the case of the composite section are:

1. The CM parameter can be assigned to I-shaped sections only. A CM (composite) section is one obtained by considering a portion of a concrete slab to act in unison with the I shaped steel section. FC is the strength or grade of concrete used in the slab. In the USA, FC is called the specified compressive strength of concrete. Typical values of FC range between 2.0 and 5.0 ksi, and 20 to 50 Mpa.

2. The width of the concrete slab (CW) (if not entered) is assumed to be the width of the top flange of the steel section + 16 times the thickness of the slab.

3. In order to calculate the section properties of the cross-section, the modular ratio is calculated assuming that:

   \[ E_s = \text{Modulus of elasticity of steel} = 29,000 \text{ ksi} \]
   \[ E_c = \text{Modulus of elasticity of concrete} = 1,802.5 \sqrt{F_c} \text{ ksi} \]

   Where \( F_c \) (in ksi) is defined earlier.

Some other general notes on this subject of member property designations are:

1. The T parameter stands for a T-shaped section obtained by cutting an I-shaped section at exactly its mid height level along the web. Hence, the area of a T shape is exactly half the area of the corresponding I shape. The depth of a T shape is half the depth of the I shape it was cut from.

   What we refer to as I shaped sections are sections which look like the English alphabet I. The American Wide Flange, the British UB and UC sections, Japanese H sections, etc., all fall under this category. Consequently, the 'T' shape cut from a Japanese H shape is one obtained by cutting the H shape at exactly its mid-height level of the web.

   Not all I shaped sections have a corresponding T. This may be inferred by going through the section libraries of individual countries and organizations. In such cases, if a user were to specify such a T section, the program will terminate with the message that the section does not exist.

2. Steel Cover plates also can be added only to I shaped sections. Thus, the TC, BC, and TB are not applicable to any shape other than an I shape.

Related Links

- G.6.2 Built-In Steel Section Libraries (on page 2325)
- M. To add a new table section property (on page 730)
- G.6.2 Built-In Steel Section Libraries (on page 2325)
- G.6.7 Composite Beams and Composite Decks (on page 2328)

TR.20.2 Prismatic Property Specification

The following commands are used to specify section properties for prismatic cross-sections.
General Format

For the PRISMATIC specification, properties are provided directly (End each line but last with a hyphen “-”) as follows:

```
property-spec = * { AX f1 | IX f2 | IY f3 | IZ f4 | AY f5 | AZ f6 | YD f7 | ZD f8 | YB f9 | ZB f10 }
```

Where:

- **AX** \( f_1 \): Cross sectional area of the member. Set to zero for TEE, Rectangular, Trapezoid, or circular.
- **IX** \( f_2 \): Torsional constant.
- **IY** \( f_3 \): Moment of inertia about local y-axis.
- **IZ** \( f_4 \): Moment of inertia about local z-axis (usually major).
- **AY** \( f_5 \): Effective shear area in local y-axis.
- **AZ** \( f_6 \): Effective shear area in local z-axis.

If any of the previous six parameters are omitted, it will be calculated from the YD, ZD, YB, and/or ZB dimensions.

- **YD** \( f_7 \): Depth of the member in local y direction. Used as the diameter of section for circular members.
- **ZD** \( f_8 \): Depth of the member in local z direction. If ZD is not provided and YD is provided, the section will be assumed to be circular.
- **YB** \( f_9 \): Depth of stem for T-section.
- **ZB** \( f_{10} \): Width of stem for T-section or bottom width for TRAPEZOIDAL section.

The values that STAAD calculates for the omitted terms can be obtained by specifying the command PRINT MEMBER PROPERTIES.

The values of many of the derived properties like shear areas (AY, AZ), section modulii (SY, SZ), etc. will be shown in the output file.

This command can be used regardless of the manner in which the properties are specified (e.g., PRISMATIC, user table, built-in table).

**Related Links**
- [G.6.1 Prismatic Properties](on page 2323)
- [M. To assign a prismatic section](on page 733)
- [D2.A.1 Section Types for Concrete Design](on page 1627)
- [D4.A.1 Section Types for Concrete Design](on page 1696)
- [D8.A.1 Section Types for Concrete Design](on page 1887)
TR.20.2.1 Prismatic Tapered Tube Property Specification

The following commands are used to specify section properties for prismatic tapered tube cross-sections. For the property types shown below, additional information can be obtained from Table 2.1 of the ASCE 72 document, 2nd edition.

**General Format**

```
property-spec = { ROUND | HEXDECCAGONAL | DODECAGONAL | OCTAGONAL | HEXAGONAL | SQUARE } START D1 END D2 THICK t
```

Where:

- **START D1** Depth of section at start of member.
- **END D2** Depth of section at end of member.
- **THICK t** Thickness of section (constant throughout the member length).

*Figure 282: Prismatic tapered tube shapes*

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Square" /></td>
<td><img src="image" alt="Hexagonal" /></td>
</tr>
<tr>
<td><img src="image" alt="Octagonal" /></td>
<td><img src="image" alt="Dodecagonal" /></td>
</tr>
</tbody>
</table>

STAAD.Pro 2467 User Manual
Notes

a. Section properties are calculated using the rules applicable for thin-walled sections.
b. Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the SET SHEAR command has no effect on the deformation computed for members with these cross sections.

Example

UNIT ...
MEMBER PROPERTY
1 PRIS ROUND STA 10 END 8 THI 0.375
2 PRIS HDC STA 15 END 10 THI 0.375
3 PRIS DOD STA 12 END 12 THI 0.375

See C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\US\Square_tapered.std and C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\US\Round_tapered.std for additional examples of designing members with prismatic tapered tube sections.

Related Links

• G.6.1 Prismatic Properties (on page 2323)
• M. To assign a prismatic section (on page 733)

TR.20.3 Tapered Member Specification

The following commands are used to specify section properties for tapered I-shapes.

General Format

\texttt{argument-list} = f1 f2 f3 f4 f5 (f6 f7)

Where:

- \texttt{f1} Depth of section at start node. Must be greater than \texttt{f3}.
- \texttt{f2} Thickness of web.
- \texttt{f3} Depth of section at end node.
**Example**

```
MEMBER PROPERTY
1 TO 5 TAPERED 15.98 0.285 11.98 6.745 .455 6.745 .455
```

See C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\US\Doubly_Symm-I_tapered.std for an additional example of designing a member with a tapered I section.

**Notes**

a. All dimensions \(f_1, f_2, \ldots, f_7\) should be in current units.

b. \(f_1\) (Depth of section at start node) should always be greater than \(f_3\) (Depth of section at end node). You must provide the member incidences accordingly.

c. Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the SET SHEAR command has no effect on the deformation computed for members with these cross sections.

**Related Links**

- [G.6.4 Tapered Sections](on page 2326)
- [M. To assign a tapered I section](on page 734)
- [G.6.4 Tapered Sections](on page 2326)
- [D1.A.5.7 Design of Web-Tapered Members](on page 1377)

**TR.20.4 Property Specification from User Provided Table**

The following commands are used to specify section properties from a previously created USER-PROVIDED STEEL TABLE.
General Format

```
member-list  UPTABLE  i1section-name
```

Where:

- **UPTABLE** stands for User-Provided TABLE
- \( i_1 \) = table number as specified previously (1 to 20)
- **section-name** = section name as specified in the table.

See [TR.20.6 Examples of Member Property Specification](on page 2472) for an example.

**Related Links**
- [G.6.3 User-Provided Steel Table](on page 2325)
- [M. To create a general section](on page 735)
- [G.6.3 User-Provided Steel Table](on page 2325)

**TR.20.5 Assign Profile Specification**

The **ASSIGN** command may be used to instruct the program to assign a suitable steel section to a frame member based on the profile-spec shown below. The current country of the member properties is used to search for a suitable size based on the profile spec provided, if a such a shape is included in that country table.

**General Format**

```
profile-spec  =  {  BEAM  |  COLUMN  |  CHANNEL  |  ANGLE  (DOUBLE)  }
```

See [TR.20.6 Examples of Member Property Specification](on page 2472) for an example.

**Note:** Sections are always chosen from the relevant built-in steel table. To find out the details of the sections that are chosen, the command **PRINT MEMBER PROPERTIES** should be provided after specification of all member properties. These commands may not work with certain tables such as cold-formed steel, Timber, Aluminum, and even some of the standard steel tables.

**Sections Provided based on Country**

The following table is a list of the actual section that is applied for each database and **ASSIGN** type.

<table>
<thead>
<tr>
<th>Country</th>
<th>Beam</th>
<th>Column</th>
<th>Channel</th>
<th>Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 US</td>
<td>W18X40</td>
<td>W14X43</td>
<td>C10X20</td>
<td>L40x406</td>
</tr>
<tr>
<td>1 BRIT</td>
<td>UB457X191X89</td>
<td>UC254X254X107</td>
<td>CH229X76</td>
<td>UA100X100X12</td>
</tr>
<tr>
<td>10 SOUTH AFRICA</td>
<td>457x75UB</td>
<td>305x198UC</td>
<td>260x90x38C</td>
<td>100x100x10L</td>
</tr>
<tr>
<td>11 SPAIN</td>
<td>IPE450</td>
<td>HEA360</td>
<td>UPN260</td>
<td>L100x100x10</td>
</tr>
<tr>
<td>12 CHINA</td>
<td>I45A</td>
<td>HW300x300</td>
<td>CH25B</td>
<td>L100x100x10</td>
</tr>
<tr>
<td>13 DUTCH</td>
<td>IPE450</td>
<td>HE360A</td>
<td>UPN260</td>
<td>L100x10</td>
</tr>
<tr>
<td>Country</td>
<td>Beam</td>
<td>Column</td>
<td>Channel</td>
<td>Angle</td>
</tr>
<tr>
<td>-------------------------</td>
<td>-------------</td>
<td>----------------</td>
<td>--------------</td>
<td>----------------</td>
</tr>
<tr>
<td>14 Aluminum</td>
<td>I12.0x12.5</td>
<td>WF8x13.0</td>
<td>CS10x6.23</td>
<td>L4x4x0.500</td>
</tr>
<tr>
<td>15 KOREA</td>
<td>I450x175x11/20</td>
<td>H350x350x12x19</td>
<td>C250x90x9/13</td>
<td>L100x10</td>
</tr>
<tr>
<td>16 VENEZUELA</td>
<td>TUB26090</td>
<td>PIP106</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>17 Cold-formed</td>
<td>(N/A)</td>
<td>(N/A)</td>
<td>10CU1.25X045</td>
<td>4LU4x060</td>
</tr>
<tr>
<td>18 MEXICO</td>
<td>IR457x112.9</td>
<td>IE305x60.7</td>
<td>CE254x22.76</td>
<td>LJ102x10</td>
</tr>
<tr>
<td>19 COLD Formed Indian</td>
<td>(N/A)</td>
<td>(N/A)</td>
<td>200CU80X5</td>
<td>100LU100X4</td>
</tr>
<tr>
<td>20 Cold Formed Indian</td>
<td>(N/A)</td>
<td>(N/A)</td>
<td>C20019</td>
<td>(N/A)</td>
</tr>
<tr>
<td>21 JOIST</td>
<td>28K10</td>
<td>(N/A)</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>22 AITC Timber</td>
<td>aspn_ST_4x12</td>
<td>aspn_ST4x4</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>23 BSCOLD</td>
<td>TU150x5</td>
<td>PP1143x6</td>
<td>150CASH50x50</td>
<td>(N/A)</td>
</tr>
<tr>
<td>24 BUTLER</td>
<td>PO8960</td>
<td>PO8718</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>25 CAN TIMBER</td>
<td>DFL_SelStr_6X14_B M</td>
<td>DFL_SelStr_10x10_B M</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>26 KINGSPAN Coldformed</td>
<td>(N/A)</td>
<td>(N/A)</td>
<td>C25010040</td>
<td>(N/A)</td>
</tr>
<tr>
<td>27 RCECO Coldformed</td>
<td>(N/A)</td>
<td>(N/A)</td>
<td>10CU1.25X045</td>
<td>4LU4x075</td>
</tr>
<tr>
<td>28 JAP Coldformed</td>
<td>BCR30030012</td>
<td>BCR30030012</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>29 Australian Cold Formed</td>
<td>200X100X9.0RHS</td>
<td>100X100X9.0SHS</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>30 Russian Cold Formed</td>
<td>PIP10X1</td>
<td>PIP10X1</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>31 STOASChM</td>
<td>B45-1</td>
<td>K35-1</td>
<td>(N/A)</td>
<td>(N/A)</td>
</tr>
<tr>
<td>32 Jindal</td>
<td>IPE450</td>
<td>HEA320</td>
<td>ISMC400x100</td>
<td>(N/A)</td>
</tr>
</tbody>
</table>
### Related Links
- 6.6.5 Assign Command (on page 2326)

### TR.20.6 Examples of Member Property Specification

This section illustrates the various options available for MEMBER PROPERTY specification.

```
UNIT ... 
MEMBER PROPERTIES
  1 TO 5 TABLE ST W8X31
  9 10 TABLE LD L40304 SP 0.25
  12 TO 15 PRISMATIC AX 10.0 IZ 1520.0
  17 18 TA ST PIPE OD 2.5 ID 1.75
  20 TO 25 TA ST TUBE DT 12. WT 8. TH 0.5
  27 29 32 TO 40 42 PR AX 5. IZ 400. IY 33. -
  39 IX 0.2 YD 9. ZD 3.
  43 TO 47 UPT 1 W10X49
  50 51 UPT 2 L40404
  52 TO 55 ASSIGN COLUMN
  56 TA TC W12X26 WP 4.0 TH 0.3
  57 TA CM W14X34 CT 5.0 FC 3.0
```

This example shows each type of member property input. Members 1 to 5 are wide flanges selected from the AISC tables; 9 and 10 are double angles selected from the AISC tables; 12 to 15 are prismatic members with no shear deformation; 17 and 18 are pipe sections; 20 to 25 are tube sections; 27, 29, 32 to 40, and 42 are prismatic members with shear deformation; 43 to 47 are wide flanges selected from the user input table number 1; 50 and 51 are single angles from the user input table number 2; 52 through 55 are designated as COLUMN members using the ASSIGN specification. The program will assign a suitable I-section from the steel table for each member.
Member 56 is a wide flange W12X26 with a 4.0 unit wide cover plate of 0.3 units of thickness at the top. Member 57 is a composite section with a concrete slab of 5.0 units of thickness at the top of a wide flange W14X34. The compressive strength of the concrete in the slab is 3.0 force/length$^2$.

**TR.20.7 Composite Decks**

A composite deck generation facility is available in the program.

**General Format**

The command syntax for defining the deck within the STAAD input file is as shown below.

```
START DECK DEFINITION
  _DECK deck-name
  PERIPHERY member-list
  DIRECTION d1 d2 d3
  COMPOSITE member-list
  OUTER member-list
  VENDOR name
  FC f1
  CT f2
  CD f3
  RBH f4
  RBW f5
  PLT f6
  PLW f7
  DIA f8
  HGT f9
  DRI f10
  SHR f11
  CMP f12
  CW f13 MEMB cw-member-list
END DECK DEFINITION
```

Where:

- **_DECK deck-name** an alphanumeric name you specify to identify the deck. The **deck-name** line must start with '_DEC' or '_DECK'. The **deck-name** is the second word and is limited to 23 characters. This name must not be the same as any group name.
The list of members belonging to the deck. TO, BY, ALL, and BEAM are permitted. ALL means all members in structure; BEAM means all beams.

**DIRECTION** $d_1 \ d_2 \ d_3$ the x, y, and z components of the direction of the deck, respectively

The following parameters may be in any order or omitted. They only apply to the composite members listed above. Do not enter a member list for these parameters.

- **FC** $f_1$: compressive strength of the concrete for all composite members listed above for this composite deck.
- **CT** $f_2$: concrete thickness
- **CD** $f_3$: concrete density
- **RBH** $f_4$: height of rib of form steel deck. This is the distance from the top of the I beam to the bottom of the concrete deck.
- **RBW** $f_5$: width of rib of form steel deck
- **PLT** $f_6$: thickness of cover plate welded to bottom flange of composite beam
- **PLW** $f_7$: width of cover plate welded to bottom flange of composite beam
- **DIA** $f_8$: diameter of shear connectors
- **HGT** $f_9$: height of shear connectors after welding
- **DR1** $f_{10}$: ratio of moment due to dead load applied before concrete hardens to the total moment
- **SHR [0 | 1]**: temporary shoring during construction.
  - 0 = no shoring
  - 1 = with shoring
- **CMP [0 | 1 | 2]**: composite action with connectors.
  - 0 = no composite action in design
  - 1 = composite action
  - 2 = ignore positive moments during design

**CW** $f_{11}$ **MEMB** $cw$-**member-list**: $f_{11}$ is the concrete width for each composite member listed. $cw$-**member-list** is the list of composite members in this deck that have this width.

Enter as many CW lines as necessary to define the width of all composite members of this deck.

This Deck definition data should be entered after the member properties have been entered.

**Notes**

1. The DECK definition must start with the START DECK DEFINITION command and end with the END command.
2. More than one DECK may be specified between the START and END.
3. The same member number may be included in up to 4 deck/groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.
4. If two or more consecutively entered decks have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.

5. The _deck-name must be unique within the Deck definitions and the Group definitions.

6. PER, DIR, OUT are data created by the GUI. Do not edit this data.

7. This Deck definition data should be entered after the member properties have been entered.

---

**Example**

```plaintext
START DECK DEFINITION
  _DECK DEC-1
  PERIPHERY 4 1640 18 38 56 50 49
  DIRECTION 0.000000 0.000000 -1.000000
  COMPOSITE 41 74 38
  OUTER 7 8 3130
  VENDOR USSTEEL
  DIA 0.700
  HGT 2.75
  CT 11.0
  FC 3.1
  RBW 2.6
  RBH 0.1
  CMP 1.0
  SHR 1
  CD 0.0000870
  CW 123.000000 MEMB41
  CW 123.000000 MEMB7
  CW 61.500000 MEMB4
  CW 61.500000 MEMB38
END DECK DEFINITION
```

---

**Related Links**

- [M. Composite Decks](on page 693)
- [G.6.7 Composite Beams and Composite Decks](on page 2328)
- [M. To add a floor load or one-way load](on page 831)
- [G.6.7 Composite Beams and Composite Decks](on page 2328)

---

**TR.20.8 Curved Member Specification**

The following commands are used to specify that a member is curved. The curve must be a segment of a circle and the internal angle subtended by the arc must be less than 180 degrees. Any non-tapered cross-section is permitted.

**General Format**

```
MEMBER CURVED member-list RADIUS r GAMMA g PRESS p
```

Where:

- `RADIUS r = radius in length units`
**GAMMA**  \( g \) = The angle in degrees used to define the plane of the circle. The angle is defined using the same set of rules used to define the orientation (beta angle) of a straight member connected between the two nodes. See Gamma Angle (on page 2477).

**PRESS**  \( p \) = Pressure/Flexibility parameter for pipe bends. See Pressure/Flexibility Parameter (on page 2476).

**Notes**

1. The radius should be in current units.
2. Certain attributes like releases, TENSION/COMPRESSION flags, and several member load types are currently not available. Section forces too are currently not available.
3. The design of curved members is not supported.

**Pressure/Flexibility Parameter**

This applies only to pipe bend (elbow) members (OD and ID entered). These members will flex more due to ovalization depending on internal pressure. The ASME Boiler and Pressure Vessel Code, Section III, NB-3687.2, 1971, for Class I components is used to calculate the flexibility reduction factor.

- Set \( p = 0 \) or omit for this flexibility increase calculation to occur with internal pressure equal to zero.
- Set \( p > 0 \) to specify internal pressure to use in this flexibility calculation. Pressure reduces the flexibility increase.
- Set \( p = -9999 \) to ignore this additional flexibility calculation and use only beam theory.
- Set \( p = \) flexibility reduction factor (-FLEXF below); which must be a negative number less than -1.0.

**ASME Pipe Elbow flexibility factors theory [ASME Section NB-3687.3]**

This section only applies if \( (\text{Bend Radius}/\text{Mean Radius}) \geq 1.70 \) or if \( (\text{Arclength}) > (2 \times \text{Mean Radius}) \)

Flexibility Factor, \( \text{FLEXF} = (1.65 \times (\text{Mean Radius})^{2})/[(t \cdot (\text{Bend Radius})] \times 1/[1 + (\text{Press})] \)

where

- \( \text{FACT} = 6 \cdot (\text{MR}/t)^{4/3} \cdot (\text{BR}/\text{MR})^{1/2} / (\text{Et}) \)
- \( \text{MR} = \) Mean Radius of elbow wall
- \( \text{BR} = \) Bend Radius
- \( \text{Press} = \) Internal Pressure
- \( t = \) elbow wall thickness
- \( E = \) Modulus of Elasticity

If the Flexibility Factor computed is less than 1.0, then STAAD.Pro will use 1.0. The Flexibility Factor directly multiplies or contributes to most non-shear terms in the elbow flexibility matrix.

**Notes**

1. The input for defining the curved member involves 2 steps. The first is the member incidence, which is the same as that for a straight line member. The second is the command described above, which indicates that the segment between the 2 nodes of the member is curved, and not a straight line.
2. Any non-tapered cross section property currently available in STAAD can be assigned to these members.
3. Currently, two load types are permitted on curved members. One is the SELFWEIGHT load type, described in TR.32.9 Selfweight (on page 2685). The other is the uniformly (UNI) distributed load type of the MEMBER LOAD options explained in TR.32.2 Member Load Specification (on page 2653). The uniformly distributed load has to be applied over the full span of the member. Other member loads such as LINEAR, TRAP, CONCENTRATED force or moment, UNIFORM moment, etc. are not supported.
4. Some of the other member load types such as PRESTRESS, TEMPERATURE, STRAIN loads, etc. are also not currently supported. These options too are expected to become available in future versions of the program.

5. The results of the analysis currently consist of the nodal displacements of the ends of the curved member, and the member end forces. The nodal displacements are in the global coordinate system. The member end forces are in the local coordinate system, with each end of the member having its own unique local axis system. Results at intermediate sections, such as sectional displacements, and sectional forces will be available in future versions of the program.

**Gamma Angle**

The plane of the circle defines the plane formed by the straight line joining the two ends of the arc, and the local Y axis of an imaginary straight member between those two points. The positive value of the GAMMA angle is obtained using the same sense as the positive value of the beta angle of that imaginary straight line member whose local Y axis points towards the vertex of the arc.

Several diagrams intended to show the GAMMA angle for various segments lying in the three global planes are shown.

*Figure 283: Gamma angle for various configurations of the circular arc lying in the global XY plane*
Figure 284: Gamma angle for various configurations of the circular arc lying in the global XY plane

Figure 285: Gamma angle for various configurations of the circular arc lying in the global YZ plane
Figure 286: Gamma angle for various configurations of the circular arc lying in the global YZ plane

Figure 287: Gamma angle for various configurations of the circular arc lying in the global XZ plane
Member local axis system

The local axis directions for curved members are dependent on the point of interest along the curve. The general rules for local axis, as laid out in G.4.2 Local Coordinate System (on page 2297) are applicable. The figure shown later for member end forces indicates the directions of axes at the start and end nodes.

Rotation of local axis

There is a limited facility available to change the orientation of a curved member cross section. The cross-section may be at the default position where the strong axis (local y) is normal to the plane of the curve and the weak axis is in that plane.

The BETA ANGLE and REFERENCE POINT options, explained in G.4.3 Relationship Between Global and Local Coordinates (on page 2301) and TR.26.2 Specifying Constants for Members and Elements (on page 2503), are not available for curved members.

Sign conventions

The displacements of the nodes of the curved member are along the global axis system just as in the case of straight members.

The member end forces for curved members are quite similar to that for straight members. The only distinguishing item is that they are normal and tangential to the local axis at the corresponding ends. For example, FX at the start is tangential to the curve at the start node, and FX at the end is tangential to the curve at the end node. Similarly, FZ is along the radial direction at the two ends.

Member releases, offsets, tension/compression, truss and cable may not be specified for curved beams.
Figure 289: Sign conventions for member end actions; Global Y is vertical

Example

STAAD SPACE
UNIT KIP FEET
JOINT COORD CYL REVERSE
1 150 0 0 13 150 0 90
REPEAT 1 30 0 0
REPEAT ALL 1 0 15 0
MEMBER INCIDENCES
1 1 27 26
101 27 28 112
113 40 41 124
201 27 40 213
START GROUP DEFINITION
MEMBER
_COLUMN 1 TO 26
_CIRCUMFERENTIAL 101 TO 124
_RADIAL 201 TO 213
END GROUP DEFINITION
MEMBER PROPERTIES
Related Links

- G.6.8 Curved Members (on page 2329)
- M. To add a curved beam (on page 646)
- G.6.8 Curved Members (on page 2329)
- G.4.3 Relationship Between Global and Local Coordinates (on page 2301)

TR.20.9 Applying Fireproofing on members

STAAD.Pro includes a method to automatically consider the weight of fireproofing material applied to structural steel.

General Format

```
MEMBER FIREPROOFING
member-list FIRE { BFP | CFP } THICKNESS f1 DENSITY f2
```

Where:

- **THICKNESS f1**  \(T\) in the figures below in length units
- **DENSITY f2**  density of fireproofing material in \((\text{force} \cdot \text{length}^{-3})\) units

In the actual load case itself, nothing besides the SELFWEIGHT command is necessary to instruct the program to include the weight of the fireproofing material in the selfweight calculation.

Two types of fireproofing configurations are currently supported. They are:

- block fireproofing
- contour fireproofing

**Block Fireproofing (BFP)**

The following figure shows this configuration. The fire-protection material forms a rectangular block around the steel section.
The area of fireproofing material \( A_{fp} \) at any section along the member length is calculated in the following manner.

For Wide Flanges (I-shaped sections), Channels and Tees,
\[
A_{fp} = [(B_f + 2T) \times (D + 2T)] - A_{steel}
\]

For single angles,
\[
A_{fp} = [(B_f + 2T) \times (D + 2T)] - A_{steel}
\]

where

- \( B_f \) = the flange width
- \( D \) = the overall depth of the steel section
- \( T \) = the thickness of the fireproofing material beyond the outer edges of the cross section as dimensioned in the next figure
- \( A_{steel} \) = area of the steel section

Where:

![Figure 290: Block fireproofing (BFP) on various shapes](image)

**Contour Fireproofing (CFP)**

In this configuration, the fire-protection material forms a coating around the steel section as shown in the next figure. The area of fireproofing material \( A_{fp} \) for this case is calculated in the following manner.

For Wide Flanges (I-shaped sections)
\[
A_{fp} = [(B_f + 2T) \times (T_f + 2T)] + [(D - 2T - 2T) \times (T_w + 2T)] - A_{steel}
\]

For single angles,
\[
A_{fp} = [(L_1 + 2T) \times (2T + T_a)] + (L_2 - T_a) \times (2T + T_a)] - A_{steel}
\]

For Tees,
\[
A_{fp} = [(B_f + 2T) \times (T_f + 2T)] + [(D - T_f) \times (T_w + 2T)] - A_{steel}
\]

where

- \( B_f \) = the flange width
- \( D \) = the overall depth of the steel section
- \( T \) = the thickness of the fireproofing material beyond the outer edges of the cross section as dimensioned in the next figure
- \( T_f \) = the thickness of the flange for the I shape and Tee
- \( T_a \) = the thickness of the leg of the angle
- \( T_w \) = the thickness of the web for the I shape and Tee
- \( A_{steel} \) = area of the steel section
The number of input items required to apply this attribute is:

- the type of fireproofing
- the thickness $T$ shown in the above figures
- the density of the fireproofing material
- the members on which it is to be applied

For each such member, $A_{fp}$ is calculated and multiplied by the density of the fireproofing material to obtain the weight per unit length of the member. This is added to the weight per unit length of the steel section itself and the total is used in calculating selfweight. Hence, SELFWEIGHT must be one of the load components of load cases if the weight of the fireproofing material should be considered as part of those load cases.

Notes

1. STAAD calculates the fireproofing weight only for the following sections:
   - For block fireproofing - I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, single channels, angles and Tees.
   - For CFP-contour fireproofing - the sections are I-beam straight or tapered, angle, Tee.
   - I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, angles and Tees.

2. Fireproofing weight is not calculated for the following section types: Pipe, tube, composite I beams with slab on top, double channel, double angle, HSS, I-beam with cover plates, prismatic, solid circle or rectangle, castellated, cold formed sections, wood, aluminum, tapered poles, etc.

Example Problem

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNIT KIP FEET</td>
</tr>
<tr>
<td>JOINT COORDINATES</td>
</tr>
<tr>
<td>1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.</td>
</tr>
<tr>
<td>MEMBER INCIDENCE</td>
</tr>
<tr>
<td>1 1 2 ; 2 2 3 ; 3 3 4</td>
</tr>
<tr>
<td>MEMBER PROPERTY AMERICAN</td>
</tr>
<tr>
<td>1 3 TABLE ST W12X26</td>
</tr>
<tr>
<td>2 TABLE ST W14X34</td>
</tr>
<tr>
<td>CONSTANTS</td>
</tr>
<tr>
<td>E STEEL ALL</td>
</tr>
<tr>
<td>POISSON STEEL ALL</td>
</tr>
<tr>
<td>DENSITY STEEL ALL</td>
</tr>
<tr>
<td>SUPPORT</td>
</tr>
</tbody>
</table>

Figure 291: Contour fireproofing (CFP) on various shapes
Concrete design specifications recommend the use of cracked section properties for the analysis and design of concrete sections. The methodology to handle cracked section properties is nonlinear in nature (i.e., the section capacities should be checked and modified depending upon the section forces the section is handling). The model should then be re-analyzed with modified reduced section properties and redesigned. This iteration should be continued until the forces in all sections designed are below the allowable limit of ultimate strength.

In STAAD.Pro, you can specify a set of reduction factors to be applied on the calculated section properties such as area, moments of inertia, and torsional constant. If you want to adopt this approach to account for cracking of concrete sections, refer to Section 10.11.1 of ACI 318 for a set of values to use for these reduction factors depending upon the nature of forces and moments the member is subjected to.

**Note:** The specifications in the AISC 13th edition manual suggest reducing the stiffness of the steel member during the analysis. In place of the MEMBER CRACKED command, the REDUCEDEI parameter may be used when the PERFORM DIRECT ANALYSIS command is used. See TR.37.5 Direct Analysis (on page 2806) for additional information.

**General Format**

The format of the command is:

```plaintext
MEMBER CRACKED (CODE IS1893 2016)
{ member-list | group-list } REDUCTION *{ RAX f1 | RIX f2 | RIY f3 | RIZ f4 }
```

Each reduction factor value, \( f1 \) through \( f4 \), should be a fraction of unity.

- **RAX \( f1 \)**: Reduction factor in the axial area. 
  \( RAX \) is *not* applicable for reduction factors for the IS1893 2016 code.

- **RIX \( f2 \)**: Reduction factor of the torsion constant (about the local x-axis).

- **RIY \( f3 \)**: Reduction factor in the moment of inertia about the local major axis (y-axis).
**RIZ f4**  Reduction factor in the moment of inertia about the local minor axis (z-axis).

This is a multiplication factor on the property value. It does not signify the amount by which the property is reduced, but, it is simply a value by which the unreduced property is multiplied. Thus, the calculated (or the user-specified value) of the property will be multiplied by the reduction factor to arrive at the value used in the analysis.

For example, a factor of 0.45 defined for RAX will mean that if the cross sectional area of the gross section is 0.8 ft$^2$, the value used in the analysis will be $0.8 \times 0.45 = 0.36$ ft$^2$.

Multiple factors can be assigned on the same line.

The reduction factor is considered only for analysis but not for design.

**Code-Specific Reduction Factors**

IS1893 2016 Clause 6.4.3 calls for a reduction of moment of inertia values by 0.35 for beams and 0.7 for columns of concrete members only while analyzing the structure for static seismic and response spectrum/linear dynamic analysis.

**Note:** Automated stiffness reduction analysis is not supported by the basic solver. STAAD.Pro Advanced is required for this feature.

Code-specific reduction factors will not affect non-concrete members. A separate stiffness matrix is generated for the use of IS1893 2016 static seismic and response spectra loads. For all other load cases, the analysis is performed using the unreduced stiffness matrix.

**Tip:** For a STAAD.Pro model containing multiple analysis commands, a different set of code-specific reduction factors may be used for each analysis command.

The CHANGE command has no affect or code-specific reduction factors.

### Example

```
MEMBER CRACKED
1 REDUCTION RAX 0.35 RIX 0.40 RIY 0.45 RIZ 0.45
```

### IS1893 2016 Example

```
START GROUP DEFINITION
MEMBER
  _BEAM 1 4
  _COLUMN 2 3 5 6
END GROUP DEFINITION
...
MEMBER CRACKED CODE IS1893 2016
  _COLUMN REDUCTION RIY 0.700 RIZ 0.700
  _BEAM REDUCTION RIY 0.350 RIZ 0.350
```

### Related Links
- [M. To assign cracked section properties to a member](on page 802)
- [M. To assign cracked section properties to a member](on page 802)
TR.21 Element/Surface Property Specification

Individual plate elements, and the Surface element need to have their thickness specified before the analysis can be performed. The commands for specifying this information are explained in this section. No similar properties are required for solid elements. However, constants such as modulus of elasticity, Poisson's Ratio, etc. are required.

**Related Links**
- *G.5 Finite Element Information* (on page 2308)
- *G.5.1 Plate and Shell Elements* (on page 2308)
- *G.5.2 Solid Elements* (on page 2319)
- *G.5.3 Surface Elements (Deprecated)* (on page 2322)

TR.21.1 Element Property Specification

This set of commands may be used to specify properties of plate finite elements. Elements of uniform or linearly varying thickness may be modeled using this command. The value of the thickness must be provided in current units.

Unlike members and plate/shell elements, no properties are required for solid elements. However, constants such as modulus of elasticity and Poisson's ratio are to be specified.

**General Format**

```
ELEMENT PROPERTY

 element-list THICKNESS f1 (f2 f3 f4)
```

Where:

- **THICKNESS f1**: thickness of the element
- **f2 ... f4**: thicknesses at other nodes of the element if different from f1

**Example**

```
UNIT ...
ELEMENT PROPERTY
1 TO 8 14 16 THI 0.25
```

**Related Links**
- *G.5.1 Plate and Shell Elements* (on page 2308)
- *M. To specify plate thickness* (on page 806)
- *G.5 Finite Element Information* (on page 2308)
- *G.5.1 Plate and Shell Elements* (on page 2308)
- *G.5.2 Solid Elements* (on page 2319)
- *G.5.3 Surface Elements (Deprecated)* (on page 2322)
TR.21.2 Surface Property Specification

This set of commands may be used to specify properties of surface entities.

Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

General Format

SURFACE PROPERTY

surface-list THICKNESS t

Where:

\( t \) = Thickness of the surface element, in current units.

Example

SURFACE PROPERTY
1 TO 3 THI 18

Related Links

- G.5.3 Surface Elements (Deprecated) (on page 2322)

TR.22 Member and Element Releases

STAAD.Pro allows specification of releases of degrees of freedom for frame members and plate elements.

Related Links

- G.7 Member and Element Release (on page 2329)

TR.22.1 Member Release Specification

This set of commands may be used to fully release specific degrees of freedom at the ends of frame members. They may also be used to describe a mode of attachment where the member end is connected to the joint for specific degrees of freedom through the means of springs.

General Format

MEMBER RELEASES

member-list {START | END | BOTH } { *{ FX | FY | FZ | MX | MY | MZ } | *(KFX f1 | KFY f2 | KMX f3 | KMY f4 | KMY f5 | KMZ f6 ) | {MP f7 | *{ MPX f8 | MPY f9 | MPZ f10 } } }

Where:

FX ... MZ represent force-x through moment-z degrees of freedom in the member local axes
Technical Reference of STAAD Commands
TR.22 Member and Element Releases

KFX \( f_1 \) … KMZ \( f_6 \)  
spring constants for these degrees of freedom, in current units

MP \( f_7 \)  
partial moment release factor for all three moments

MPX \( f_8 \), MPY \( f_9 \), MPZ \( f_{10} \)  
partial moment release factors for each moment separately. The moment related stiffness coefficient will be multiplied by a factor of \( (1 - f_n) \) at the specified end. Release factors must be in the range of 0.001 through 0.999.

Note: If FX through MZ is used, it signifies a full release for that degree of freedom and if KFX through KMZ is used, it signifies a spring attachment.

Notes

a. Member releases are a means for describing a type of end condition for members when the default condition, namely, fully moment and force resistant, is not applicable. Examples are bolted or riveted connections. Partial moment releases are a way of specifying bending and torsional moment capacity of connections as being a fraction of the full bending and torsional strength.

b. It is important to note that the factor \( f_1 \) indicates a reduction in the stiffness corresponding to the rotational degrees of freedom MX, MY, and MZ. In other words, you should not expect the moment on the member to reduce by a factor of \( f_1 \). It may be necessary to perform a few trials in order to arrive at the right value of \( f_1 \) that results in the desired reduction in moment.

c. The START and END are based on the MEMBER INCIDENCE specification. The BOTH specification will apply the releases at both ends.

d. At any end of the member—for any particular DOF—full, partial, and spring release cannot be applied simultaneously. Only one out of the three is permitted.

e. If MY (or MZ) is fully released at both ends, then VZ (or VY) cannot be transmitted through the member. The final shears in the member will be entirely due to loads applied directly to the member.

Example

<table>
<thead>
<tr>
<th>MEMBER RELEASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 3 TO 9 11 12 START KFX 1000.0 MY MZ</td>
</tr>
<tr>
<td>1 10 11 13 TO 18 END MZ KMX 200.0</td>
</tr>
</tbody>
</table>

In this example, for members 1, 3 to 9, 11 and 12, the moments about the local Y and Z axes are released at their start joints (as specified in MEMBER INCIDENCES). Further, these members are attached to their START joint along their local x axis through a spring whose stiffness is 1000.0 units of force/length. For members 1, 10, 11 and 13 to 18, the moment about the local Z axis is released at their end joint. Also, the members are attached to their END joint about their local x axis through a moment-spring whose stiffness is 200.0 units of force-length/Degree. Members 1 and 11 are released at both start and end joints, though not necessarily in the same degrees of freedom.

Partial Moment Release

Moments at the end of a member may be released partially using the MP option (to provide the same partial release in all. This facility may be used to model partial fixity of connections. The following format may be used to provide a partial moment release. This facility is provided under the MEMBER RELEASE option and is in addition to the existing RELEASE capabilities.
MEMBER RELEASE
15 TO 25 START MP 0.25

The above RELEASE command will apply a factor of 0.75 on the moment related stiffness coefficients at start node of members 15 to 25.

Related Links
- G.7 Member and Element Release (on page 2329)
- M. To assign member end release (on page 799)
- M. To assign specifications to physical members (on page 804)

TR.22.2 Element Release Specification

This set of commands may be used to release specified degrees of freedoms at the corners of plate finite elements.

General Format

```
ELEMENT RELEASE

element-list { J1 | J2 | J3 | J4 } *{ FX | FY | FZ | MX | MY | MZ }
```

Where:

- J1, J2, J3 and J4 = signify joints in the order of the specification of the element incidence. For example, if the incidences of the element were defined as 35 42 76 63, J1 represents 35, J2 represents 42, J3 represents 76, and J4 represents 63.
- FX through MZ = represents forces/moments to be released per local axis system.

**Note:** Element releases at multiple joints cannot be specified in a single line. Those must be specified separately as shown below.

Examples

**Example of Correct Usage**

```
ELEMENT RELEASE
10 TO 50 J1 MX MY
10 TO 50 J2 MX MY
10 TO 50 J3 MY
10 TO 50 J4 MY
```

**Example of Incorrect Usage**

```
ELEMENT RELEASE
10 TO 50 J1 J2 MX MY
10 TO 50 J3 J4 MY
```
Notes

a. All releases are in the local axis system. See Figure 1.13 (on page 2308) for the various degrees of freedom. Fx and Fy have the same sense as Sx and Sy in Figure 1.13 (on page 2308). Fz has the same sense as SQx or SQy. Generally, do not over release. The element must still behave as a plate after the releases.

b. Selfweight is applied at each of the nodes as if there were no releases.

c. Thermal stresses will include the fixed-end thermal pre-stress as if there were no release.

d. May not be used with the Element Plane Stress or Element Ignore Inplane Rotation commands on the same element.

e. Note that the usual definitions of local Mx and My are reversed here. See Figure 1.13 (on page 2308) for the definitions of Mx and My. Releasing Fz, Mx, My will release all bending capability. Releasing Fx, Fy, Mz will release all in-plane stiffness.

Related Links

- G.7 Member and Element Release (on page 2329)
- M. To assign plate corner release (on page 807)
- G.5.1 Plate and Shell Elements (on page 2308)

TR.22.3 Element Ignore Stiffness

Structural units like glass panels or corrugated sheet roofs are subjected to loads like wind pressures or snow loads. While these units are designed to carry those loads and transmit those loads to the rest of the structure, they are not designed to provide any additional stiffness to the structure. One way to handle the situation is to not input the unit as part of the structural model, and apply the load using load generation techniques like AREA LOAD or FLOOR LOAD.

STAAD provides another way of handling such units. This is through the help of the ELEMENT IGNORE STIFFNESS command. To use this feature, the glass panel or roof unit must be defined using plate elements. The IGNORE STIFFNESS command enables one to consider the unit just for the purpose of application of ELEMENT LOAD commands, while its stiffness will not be considered during the assembly of the stiffness matrix. In other words, it is like an INACTIVE member which is active for ELEMENT LOAD command application but INACTIVE for stiffness. Like the INACTIVE command, the plates listed here will become active at the next CHANGE command. To keep them inactive, re-enter this data after each CHANGE.

The SELFWEIGHT, ELEMENT WEIGHT, TEMPERATURE, and mass of the plates listed here will still be ignored.

General Format

IGNORE STIFFNESS {ELEMENT} element-list

Example

IGNORE STIFFNESS ELEMENT 78 TO 80

Related Links

- G.5.1 Plate and Shell Elements (on page 2308)
- M. To ignore plate stiffness (on page 810)
TR.23 Axial Member Specifications

A member can have only one of the following specifications:

- Truss
- Tension-only
- Compression-only
- Cable

If multiple specifications are applied to the same member, only the last entered will be used (Warnings will be printed).

**Note:** MEMBER TRUSS, MEMBER TENSION, MEMBER COMPRESSION, and MEMBER CABLE are axial-only for stiffness. MEMBER CABLE are special truss members that may also be specified as tension-only.

**TR.23.1 Member Truss Specification**

This command may be used to model a specified set of members as TRUSS members.

This specification may be used to specify TRUSS type members in a PLANE, SPACE or FLOOR structure. The TRUSS members are capable of carrying only axial forces. Typically, bracing members in a PLANE or SPACE frame will be of this nature.

**General Format**

```plaintext
MEMBER TRUSS member-list (TENSION f1)
```

Where:

- **TENSION f1** optional initial tension in the truss member, in current units

**Note:** For Nonlinear CABLE ANALYSIS Only: The Tension parameter is ignored except in Nonlinear Cable Analysis. For that analysis type, a truss with pretension is considered to be nonlinear (large displacement). In this analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command.

This command is superfluous when a TRUSS type structure has already been specified using the command STAAD TRUSS.

**Example**

```plaintext
MEMB TRUSS
1 TO 8 10 12 14 15
```
Notes

a. The TRUSS member has only one degree of freedom—the axial deformation. Note also that Member Releases are not allowed. Selfweight and transverse loads may induce shear/moment distributions in the member.

b. Member loads are lumped at each end, whereas a frame member with moment releases only retains the axial component of the applied member load.

Related Links

• G.8.1 Truss and Tension- or Compression-Only Members (on page 2330)
• M. To assign axial action members (on page 798)
• M. To assign specifications to physical members (on page 804)
• G.8.1 Truss and Tension- or Compression-Only Members (on page 2330)
• G.9 Tension- and Compression- Only Springs (on page 2336)
• G.8.2.1 Linearized Cable Members (on page 2331)
• G.8.2.2 Nonlinear Cable and Truss Members (on page 2332)
• G.17.2.7 Nonlinear Cable or Truss Analysis (on page 2357)

TR.23.2 Member Cable Specification

This command may be used to model a specified set of members as cable members.

Cable members, in addition to elastic axial deformation, are also capable of accommodating the stiffness effect of initial tension and tension due to static loads. When used in a nonlinear cable analysis, cable members are capable of accommodating large displacements. See G.8.2 Cable Members (on page 2330) for a theoretical discussion of cable members.

General Format

```
MEMBER  CABLE

member-list  cable-spec

cable-spec  = { TENSION f1 ( { START | END } ) | LENGTH f2 } *[ (FWX f3) | (FWY f4) | (FWZ f5) ]
```

Where:

- **TENSION f1**: Initial tension in cable member (in current units), when the TENSION option is specified.
- **LENGTH f2**: Unstressed cable length (in current units), when the initial LENGTH option is specified.
- **FWX f3, FWY f4, FWZ f5**: Multiplying factors on self weight components applied in the global X, Y, and Z directions, respectively.
- **START, END**: The initial tension is measured at the cable start node or cable end node, respectively. This is used for advanced cable analysis (refer to Note 5 in Nodes for use with Advanced Cable Analysis).

Notes for use with Standard Cable Analysis

1. The tension specified in the cable member is applied on the structure as an external load as well as is used to modify the stiffness of the member. The tension value must be positive to be treated as a cable otherwise it is...
**Technical Reference of STAAD Commands**

**TR.23 Axial Member Specifications**

A truss (See G.8.2 Cable Members [on page 2330]). If the TENSION parameter or the value is omitted, a minimum tension will be used.

The end at which initial tension is measured is not used for standard cable analysis (i.e., START or END is ignored).

2. This is a truss member but not a tension-only member unless you also include this member in a MEMBER TENSION input (See TR.23.3 Member Tension/Compression Specification [on page 2495]). Note also that Member Releases are not allowed.

3. The tension is a preload and will not be the final tension in the cable after the deformation due to this preload.

4. The tension is used to determine the unstressed length. That length will be shorter than the distance between the joints by the distance that the tension will stretch the cable.

5. No weight (other than the assumed self weight) is used with standard cable analysis.

6. Cables should not be used in analysis involving frequency extraction or for dynamic loading conditions such as response spectrum, time history, or steady state.

**Notes for use with Advanced Cable Analysis**

1. A cable member is a truss member that has three translational degrees of freedom only. Note also that member releases are not allowed for a cable member.

2. FWY is used to add any additional weight that may be acting on the cable along with self weight before the application of external applied load. This additional weight will be added to cable self-weight in global Y direction. These are used to find the initial cable profile before the application of the external load.

3. If no self weight is included in the external applied load and also FWY parameter is not defined, the program will calculate self weight of the cable member and include it in the analysis. Warning message will be issued in the output file.

4. If self weight is included in the external applied load, during calculation cable self weight will not be considered in the external applied load vector. The reason is that it will be considered separately while finding the initial cable configuration under self weight and thus cannot be considered as external applied load.

5. If START or END is not specified with the initial TENSION parameter in the cable member, then the average tension is assumed. The average option is suitable only for a taut cable. This is because in a taut cable, the undeformed length is shorter than the chord length. Due to this, the taut cable carries significant tension with little sag. The undeformed length, Lu, is calculated as:

\[
L_u = \frac{L_c}{1 + \frac{T_{avg}}{EA}}
\]

where

- \(L_c\) = chord length
- \(T_{avg}\) = initial average tension in the taut cable
- \(E\) = Young’s modulus
- \(A\) = area of the cross-section

However, for a slack cable, this does not hold true because the undeformed length is longer than the chord length and the cable member has significant sag. The catenary curve effect is required to be included. Thus, for a slack cable, initial TENSION should be defined either at the START or END node of the cable member. For a cable member which is considered to have significant sag, defining the average tension is not a suitable option.

6. Cables should not be used in analysis involving frequency extraction or for dynamic loading conditions such as response spectrum, time history, or steady state.
Example

A series of cable members with a tension of 15.5 specified for use with either standard or advanced cable analysis:

```
MEMB CABLE
20 TO 25 TENSION 15.5
```

A series of cable members with a tension of 15.5 at the cable start node for use with advanced cable analysis. An additional weight on the cable of 20% of the cable self weight which acts in the same direction of the cable's self weight is used.

```
MEMB CABLE
20 TO 25 TENSION 15.5 START FWY -0.2
```

Related Links

- [M. To assign nonlinear cable members](#) (on page 801)
- [G.8.2.1 Linearized Cable Members](#) (on page 2331)
- [G.8.2.2 Nonlinear Cable and Truss Members](#) (on page 2332)
- [G.17.2.7 Nonlinear Cable or Truss Analysis](#) (on page 2357)
- [G.8.2.3 Nonlinear Cable Members for Advanced Cable Analysis](#) (on page 2332)
- [G.17.2.8 Advanced Nonlinear Cable Analysis](#) (on page 2358)

TR.23.3 Member Tension/Compression Specification

This command may be used to designate certain members as Tension-only or Compression-only members. Tension-only members are truss/cable members that are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them.

General Format

```
MEMBER TENSION

member-list
```

or

```
MEMBER COMPRESSION

member-list
```

or

```
MEMBER TENSION 0
```

Linear Tension/Compression Analysis

Compression-only members are truss members that are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them. Member Releases are not allowed on members with this attribute.

The procedure for analysis of Tension-only or Compression-only members requires iterations for every load case and therefore may be quite involved, you may also consider using the INACTIVE specification (instead of Tension/Compression) if the solution time becomes unacceptably high.
If a CHANGE command is used (because of a change in the list of tension members, cable tension, supports, etc.), then the SET NL command must be used to convey to STAAD that multiple analyses and multiple structural conditions are involved.

**Note:** For Nonlinear cable analysis, this command is unnecessary and ignored. Cables are automatically assumed to be partially to fully tension only (except that there should always be selfweight) without this command. In this analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command.

**a.** Loads that have been defined on members declared as MEMBER TENSION or MEMBER COMPRESSION will be active even when the member becomes INACTIVE during the process of analysis. This applies to SELFWEIGHT, MEMBER LOADS, PRESTRESS & POSTSTRESS LOADS, TEMPERATURE LOAD, etc.

**b.** A member declared as a TENSION only member or a COMPRESSION only member will carry axial forces only. It will not carry moments or shear forces. In other words, it is a truss member.

**c.** Do not use Load Combination to combine these cases. Tension/Compression cases are Nonlinear and should not be linearly combined as in Load Combination. Use a primary load case with the Repeat Load command.

### Example

<table>
<thead>
<tr>
<th>MEMBER TENSION</th>
<th>12 17 19 TO 37 65</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER COMPRESSION</td>
<td>5 13 46 TO 53 87</td>
</tr>
</tbody>
</table>

### Member Tension 0

This command switches off all tension/compression only specifications for load cases which are specified subsequent to this command, usually entered after a CHANGE command. There is no list associated with this command. Hence, for any further primary load cases, the tension/compression only attributed is disabled for all members.

### Example

The following is the general sequence of commands in the input file if the MEMBER TENSION or MEMBER COMPRESSION command is used. This example is for the MEMBER TENSION command. Similar rules are applicable for the MEMBER COMPRESSION command. The dots indicate other input data items.

```
STAAD ...
SET NL ...
UNITS ...
JOINT COORDINATES ...
MEMBER INCIDENCES ...
ELEMENT INCIDENCES ...
CONSTANTS ...
MEMBER PROPERTY ...
```
SUPPORTS
...
MEMBER TENSION
...
LOAD 1
...
LOAD 2
...
LOAD 3
...
LOAD 4
...
LOAD 5
REPEAT LOAD
...
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ...
PRINT ...
PARAMETER
...
CHECK CODE ...
FINISH

Notes

a. See TR.5 Set Command Specification (on page 2413) for explanation of the SET NL command. The number that follows this command is an upper bound on the total number of primary load cases in the file.
b. STAAD performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a SET ITERLIM i command (i > 10) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high.
c. The principle used in the analysis is the following.
   • The program reads the list of members declared as MEMBER TENSION and/or COMPRESSION.
   • The analysis is performed for the entire structure and the member forces are computed.
   • For the members declared as MEMBER TENSION / COMPRESSION, the program checks the axial force to determine whether it is tensile or compressive. If the member cannot take that load, the member is "switched off" from the structure.
   • The analysis is performed again without the switched off members.
   • Up to 10 iterations of the above steps are made for each load case, unless a higher value is set using the command ITERLIM.
   • This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable.
d. A revised MEMBER TENSION / COMPRESSION command and its accompanying list of members may be provided after a CHANGE command. If entered, the new MEMBER TENSION/COMPRESSION commands replace all prior such commands. If these commands are not entered after a CHANGE, then the previous commands will still be applicable.
e. The MEMBER TENSION command should not be used if the following load cases are present: Response Spectrum load case, Time History Load case, or Moving Load case. If used, the MEMBER TENSION / COMPRESSION will be ignored in all load cases.
If UBC Load cases are included, then follow each UBC load case with an Analysis command, then a Change command.

If the model requires a PDelta analysis, then only the first method as described in TR.37.2 P-Delta Analysis Options (on page 2797) (i.e., small or large delta) may be used.

Related Links
- G.8.1 Truss and Tension-or Compression-Only Members (on page 2330)
- M. To assign axial action members (on page 798)
- G.8.1 Truss and Tension-or Compression-Only Members (on page 2330)

TR.24 Element Plane Stress and Ignore Inplane Rotation Specification

These commands allow the user to model the following conditions on plate elements

a. PLANE STRESS condition
b. In-plane rotation stiffness reformulated to be rigid or to be zero.

General Format

```
ELEMENT { PLANE STRESS | RIG ID ( INPLANE ROTATION ) | IGNORE ( INPLANE ROTATION) }
```

The PLANE STRESS specification allows the user to model selected elements for PLANE STRESS action only [No bending or transverse shear stiffness].

The RIGID INPLANE ROTATION command causes the program to connect the corner Mz "in-plane rotation" action to the other corner Mz rotations rigidly. The STAAD plate element formulation normally produces a very soft Mz stiffness that improves the inplane shear deformation. However when the plate Mz is connected to a beam bending moment as the only load path for that moment, then the RIGID INPLANE option may be used to have that element carry the Mz moment to the other joints rigidly to avoid the instability at the beam end. Usually only the elements connected to beams in this manner would have this specification.

The IGNORE INPLANE ROTATION command causes the program to ignore "in-plane rotation" actions. The STAAD plate element formulation normally includes this important action automatically. However, it may be noted that some element formulations ignore this action by default. The user may utilize this option to compare STAAD results with solutions from these programs.

These options are exclusive of each other and also exclusive of element releases. No single element may have more than one of these options.

Example

```
ELEMENT PLANE STRESS
  1 TO 10 15 20 25 35
ELEMENT IGNORE
  30 50 TO 55
```

Related Links
- G.5.1 Plate and Shell Elements (on page 2308)
- M. To assign plates as plane stress (on page 808)
TR.25 Member Offset Specification

This command may be used to rigidly offset a frame member end from a joint to model the offset conditions existing at the ends of frame members.

General Format

```
MEMBER OFFSETS
member-list { START | END } ( LOCAL ) dx dy dz
```

Where:

- `dx`, `dy`, and `dz` correspond to the distance, measured in localized or global coordinate system, from the joint (START or END as specified) to the centroid of the starting or ending point of the members listed.
- `LOCAL` optional parameter, if not entered then `dx`, `dy`, `dz` are assumed to be in global. LOCAL means that the distances `dx`, `dy`, `dz` are in the same member coordinate system that would result if the member were not offset and BETA = 0.0.

Description

The MEMBER OFFSET command can be used for any member whose starting or ending point is not concurrent with the given incident joint. This command enables the user to account for the secondary forces, which are induced due to the eccentricity of the member. Member offsets can be specified in any direction, including the direction that may coincide with the member x-axis.
In the figure above, WP refers to the location of the centroid of the starting or ending point of the member.

**Example**

```plaintext
MEMBER OFFSET
1 START 7
1 END -6
2 END -6 -9
```

**Notes**

a. If a MEMBER LOAD (See TR.32.2 Member Load Specification (on page 2653)) is applied on a member for which MEMBER OFFSETS have been specified, the location of the load is measured not from the coordinates of the starting joint. Instead, it is measured from the offset location of the starting joint.

b. START and END is based on the user’s specification of MEMBER INCIDENCE for the particular member.

c. Local offsets are defined in the local axes prior to rotation when a BETA angle is used.

**Related Links**

- G.11 Member Offsets (on page 2334)
- M. To assign member end offsets (on page 800)
- M. To assign specifications to physical members (on page 804)
- G.11 Member Offsets (on page 2334)
TR.26 Specifying and Assigning Material Constants

Material constants are attributes like Modulus of Elasticity and Density which are required for operations like generating the stiffness matrix, computing selfweight, and for steel and concrete design.

In STAAD, there are two ways in which this data may be specified:

a. A two-step process that involves:
   a. Creating the material data by defining MATERIAL tags specified under the heading DEFINE MATERIAL (See TR.26.1 Define Material (on page 2501))
   b. Assigning them to individual members, plates and solids under the heading CONSTANTS (See TR.26.2 Specifying Constants for Members and Elements (on page 2503))

This will create commands as shown below:

```
Part 1DEFINE MATERIAL
  ... name
  ...
  END MATERIAL

Part 2CONSTANTS
  MATERIAL name ...
```

b. Assign material attributes explicitly by specifying the individual constants (See TR.26.2 Specifying Constants for Members and Elements (on page 2503)).

```
CONSTANTS
  E ...
  POISSON .
```

See TR.26.3 Surface Constants Specification (on page 2508) for an explanation for the commands required to assign material data to Surface elements.

Related Links
- G.12 Material Properties (on page 2334)

TR.26.1 Define Material

This command may be used to specify the material properties by material name. You will then assign the members and elements to this material name in the CONSTANTS command (See TR.26.2 Specifying Constants for Members and Elements (on page 2503) for details and an example).

**Note:** ISOTROPIC materials can be assigned to all element types, 2DORTHOTROPIC materials should only be assigned to plate elements.

**General Format**

```
DEFINE MATERIAL
```
then

<table>
<thead>
<tr>
<th>ISOTROPIC</th>
<th>name</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>f1</td>
</tr>
<tr>
<td>G</td>
<td>f3</td>
</tr>
<tr>
<td>POISSON</td>
<td>f6</td>
</tr>
<tr>
<td>DENSITY</td>
<td>f7</td>
</tr>
<tr>
<td>ALPHA</td>
<td>f8</td>
</tr>
<tr>
<td>DAMPING</td>
<td>f10</td>
</tr>
</tbody>
</table>

or

<table>
<thead>
<tr>
<th>2DORTHOTROPIC</th>
<th>name</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>f1</td>
</tr>
<tr>
<td>G</td>
<td>f3</td>
</tr>
<tr>
<td>POISSON</td>
<td>f6</td>
</tr>
<tr>
<td>DENSITY</td>
<td>f7</td>
</tr>
<tr>
<td>ALPHA</td>
<td>f8</td>
</tr>
<tr>
<td>DAMPING</td>
<td>f10</td>
</tr>
</tbody>
</table>

Repeat ISOTROPIC or 2DORTHOTROPIC name and values for as many materials as desired then:

END MATERIAL (DEFINITION)

Where:

- **name**: material name (name of up to 36 characters)
- **f1, f2**: specifies Young’s Modulus (E). (f2 in local Y for 2DORTHOTROPIC materials)
- **f3, f4, f5**: specifies Shear Modulus (G). For plates, the following are G values in local directions: f3 is the G for in-plane shear; f4 is the G for transverse shear in the local Y-Z direction; f5 is the G for transverse shear in the local Z-X direction. (Enter only for beams when the Poisson ratio is not in the range of 0.01 to 0.499.)
- **f6**: specifies Poisson’s Ratio. If G is not entered, then this value is used for calculating the Shear Modulus (G = 0.5xE/(1+POISSON)). This value must be in the range of 0.01 to 0.499. Poisson’s ratio must be entered for orthotropic plates or when Poisson cannot be computed from G.
- **f7**: specifies weight density
- **f8, f9**: Coefficient of thermal expansion. (f9 in local Y for 2DORTHOTROPIC materials)
- **f10**: the damping ratio to be used in computing the modal damping by the composite damping method in a dynamic analysis when CDAMP has been specified. Damping must be in the range of 0.001 to 0.990.

Note:

Any material property which you do not explicitly specify is assumed to be the default value.

- f1 defaults to 0.0 but a positive value must be entered or an error will ensue
- f2 defaults to f1
• $f_3$ defaults to $0.5x/E/(1+\text{POISSON})$
• $f_4$ defaults to $f_3$
• $f_5$ defaults to $f_4$.
• $f_6$ defaults to a sliding scale value based on $E$ (that is: 0.30 when $E$ is near that of steel, 0.33 when $E$ is near that of aluminum, 0.17 when $E$ is near that of concrete).
• $f_7$ defaults to 0.0
• $f_8$ defaults to 0.0
• $f_9$ defaults to $f_8$
• $f_{10}$ defaults to 0.0

**Tip:** If one or more of the material properties is not explicitly specified, the results may be unpredictable or even incorrect with respect to the intended behavior. Therefore, it is best practice to always specify each material property for each defined material.

**Related Links**
- [M. To create a material definition](on page 788)
- [M. To create an orthotropic material](on page 791)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3.1 Composite Damping](on page 2367)
- [G.4 Coordinate Systems and Structure Geometry](on page 2295)
- [G.4.1 Global Coordinate System](on page 2296)
- [G.4.2 Local Coordinate System](on page 2297)
- [G.4.3 Relationship Between Global and Local Coordinates](on page 2301)

**TR.26.2 Specifying Constants for Members and Elements**

This command may be used to specify the individual material properties (Modulus of Elasticity, Poisson's ratio, Density, Coefficient of linear expansion, and material damping) of the members and elements. In addition, this command may also be used to specify the member orientation (BETA angle or reference point/vector).

**General Format**

```plaintext
CONSTANTS
```
To define the orientation:

```
BETA { f5 | ANGLE | RANGLE } MEMBER memb/elem-list
{ REF f8 f9 f10 | REFJT f11 | REFVECTOR f12 f13 f14 } MEMBER memb/elem-list
```

To define the material properties:

```
MATERIAL name { MEMBER member/element-list | (ALL) }
or
{ E f1 | G f2 | POISSON f3 | DENSITY f4 | ALPHA f6 | CDAMP f7 } { MEMBER memb/elem-list | BEAM | PLATE | SOLID | (ALL) }
```

Where:

- **name** = material name as specified in the `DEFINE MATERIAL` command (Refer to TR.26.1 Define Material on page 2501).

```
{ REF f8 f9 f10 | REFJT f11 | REFVECTOR f12 f13 f14 } MEMBER memb/elem-list
```

Where:

- **memb/elem-list** = MEM, BEA, PLA, SOL, ALL. Only MEM may be followed by a list. If none are specified, the default is to use ALL, which means all members and elements; BEA means all members; PLA, all plates; SOL, all solids.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>f1</td>
<td>Specifies the modulus of elasticity, E (i.e., Young's Modulus). This value must be provided before the POISSON for each member/element in the Constants list.</td>
</tr>
<tr>
<td>G</td>
<td>f2</td>
<td>Specifies Shear Modulus (G). Enter only for beams when Poisson would not be 0.01 to 0.499.</td>
</tr>
<tr>
<td>POISSON</td>
<td>f3</td>
<td>Specifies Poisson's Ratio, ( \nu ). This value is used for calculating the Shear Modulus (( G = 0.5xE/(1+\nu) )).</td>
</tr>
<tr>
<td>DENSITY</td>
<td>f4</td>
<td>Specifies weight per unit volume, ( \gamma )</td>
</tr>
<tr>
<td>ALPHA</td>
<td>f6</td>
<td>Coefficient of thermal expansion, ( a )</td>
</tr>
<tr>
<td>CDAMP</td>
<td>f7</td>
<td>Damping ratio to be used in computing the modal damping by the composite damping method in a dynamic analysis when CDAMP has been specified. Damping must be in the range of 0.001 to 0.990.</td>
</tr>
<tr>
<td>BETA</td>
<td>f5</td>
<td>Specifies member rotation angle in degrees (Refer to G.4.3 Relationship Between Global and Local Coordinates on page 2301).</td>
</tr>
</tbody>
</table>

The following values are used in various methods to define the BETA angle based on geometry:

- **f8, f9, f10** = Global X, Y, and Z coordinates for the reference point
f11 = use location of joint f11 for the reference point, from which the BETA angle will be calculated by STAAD.Pro.

f12, f13, f14 = Establishes a Reference Vector along which the local y-axis is aligned. From the start node of the member, move by a distance of f12 along the beam’s local x axis, f13 along the local Y axis, and f14 along the local Z axis to define the end node of the reference vector. (The BETA angle is thus the angel between the local y-axis and the reference vector)

Using BETA ANGLE and RANGLE

Single angle sections are oriented according to their principal axes by default. If it is necessary to orient them such that their legs are parallel to the global axes, the BETA specification must be used. STAAD.Pro offers the following additional specifications for this purpose:

- BETA ANGLE
- BETA RANGLE

Both of the above options will result in an orientation with the legs parallel to the global axis. The ANGLE option rotates the section counterclockwise by the angle, α (where α = angle between the principal axis system and the geometric axis system of the angle). The RANGLE option rotates the section counterclockwise by an angle equal to (180° - α). For unequal angles, the right option must be used based on the required orientation.

Table 233: Effect of BETA ANGLE and BETA RANGLE commands

<table>
<thead>
<tr>
<th>BETA value</th>
<th>Zero (0)</th>
<th>ANGLE</th>
<th>RANGLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST Angle</td>
<td><img src="Diagram1.png" alt="Diagram" /></td>
<td><img src="Diagram2.png" alt="Diagram" /></td>
<td><img src="Diagram3.png" alt="Diagram" /></td>
</tr>
<tr>
<td>RA Angle</td>
<td><img src="Diagram4.png" alt="Diagram" /></td>
<td><img src="Diagram5.png" alt="Diagram" /></td>
<td><img src="Diagram6.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

**Note:** The figures in the preceding table are for a Y-UP system where the local x-axis goes into the screen. Global Y-axis is vertical. Refer to G.4.2 Local Coordinate System (on page 2297) for the orientation of angles when Z-UP is specified.
Figure 293: Orientation of an angle profile vertically aligned (i.e., local x = Global Y) with (A) BETA = 0 (default) and (B) BETA = ANGLE

**Note:** The figures is for a Y-UP system where the local x-axis goes into the into the screen. Refer to G.4.2 Local Coordinate System (on page 2297) for the orientation of angles when Z-UP is specified.

### Built-In Material Constants

For E, G, POISSON, DENSITY, ALPHA, and CDAMP, built-in material names can be entered instead of a value for \( f_1 \). The built-in names are STEEL, CONCRETE, & ALUMINUM. Appropriate values will be automatically assigned for the built-in names.

**Table 234: Constants (in Kip, inch, Fahrenheit units)**

<table>
<thead>
<tr>
<th>Constant</th>
<th>Material</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Steel</td>
<td>Concrete</td>
</tr>
<tr>
<td>E (US)</td>
<td>29,000</td>
<td>3,150</td>
</tr>
<tr>
<td>Poisson’s</td>
<td>0.30</td>
<td>0.17</td>
</tr>
<tr>
<td>Density</td>
<td>0.000283</td>
<td>0.000868</td>
</tr>
<tr>
<td>Alpha</td>
<td>6.5E-6</td>
<td>5.5E-6</td>
</tr>
<tr>
<td>CDAMP</td>
<td>0.03</td>
<td>0.05</td>
</tr>
<tr>
<td>E (nonUS)</td>
<td>29,732.736</td>
<td></td>
</tr>
</tbody>
</table>
Table 235: Constants (in MKS, Celsius units)

<table>
<thead>
<tr>
<th>Constant</th>
<th>Material</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Steel</td>
<td>Concrete</td>
</tr>
<tr>
<td>E (US)</td>
<td>199,947,960</td>
<td>21,718,455</td>
</tr>
<tr>
<td>Poisson's</td>
<td>0.30</td>
<td>.17</td>
</tr>
<tr>
<td>Density</td>
<td>76.819 541</td>
<td>23.561612</td>
</tr>
<tr>
<td>Alpha</td>
<td>12.0E-6</td>
<td>10.0E-6</td>
</tr>
<tr>
<td>CDAMP</td>
<td>.03</td>
<td>.05</td>
</tr>
<tr>
<td>E (nonUS)</td>
<td>205,000,000</td>
<td></td>
</tr>
</tbody>
</table>

Note: E (US) is used if US codes were installed or if Member Properties American is specified for an analysis; otherwise E (nonUS) is used.

Example 1

DEFINE MATERIAL ISOTROPIC CFSTEEL
E 28000.
POISSON 0.25
DENSITY 0.3E-3
ALPHA 11.7E-6
DAMP 0.075
END MATERIAL
CONSTANTS
MATERIAL CFSTEEL MEMB 1 TO 5
CONSTANTS
E 2.1E5 ALL
BETA 45.0 MEMB 5 7 TO 18
DENSITY STEEL MEMB 14 TO 29
BETA 90 MEMB X

Example 2

The REFVECTOR command is used as in the following example:

REFVECTOR 0 2 1 MEMBER 27 TO 32

This command will set BETA as 90° for all members parallel to the X-axis and instructs the program to do the following procedure:

1. Establish the beam’s local X,Y and Z axis corresponding to Beta = 0
2. Set the start node of the reference vector to be the same as the start node of the member.
3. From the start node of the reference vector, move by a distance of 0 along the beam's local X axis, 2 along the local Y axis, and 1 along the local Z axis. This establishes the end node of the reference vector.

4. At the end of step 3, the start node as well as the end node of the reference vector are known. That is the now the final direction of the member's local Y axis.

Since the local Y axis corresponding to Beta 0 is known, and the local-Y axis corresponding to the beam's final position has been established in step 4, Beta angle is calculated as the angle between these two vectors.

In this example, the angle is $\tan^{-1}(1/2) = 26.5651$ degrees

Notes

a. The value for E must always be given first in the Constants list for each member/element.
b. All numerical values must be provided in the current units.
c. It is not necessary or possible to specify the units of temperature or ALPHA. You must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for temperature (see TR.32.6 Temperature Load Specification for Members, Plates, and Solids (on page 2682)).
d. If G is not specified, but Poisson (ν) is specified, G is calculated as $E/[2(1+\nu)]$.
e. If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
f. If G and Poisson are both specified, the input value of G is used. That is, G is not calculated in this situation.
g. If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated as $((E/2G)−1)$.
h. To obtain a report of the values of these terms used in the analysis, specify the command PRINT MATERIAL PROPERTIES.
i. Local offsets are defined in the local axes prior to rotation when a BETA angle is used.

Related Links

- GS. To change the system units (on page 40)
- M. To align a member to a reference point (on page 796)
- G.4.3 Relationship Between Global and Local Coordinates (on page 2301)
- M. To assign a member rotation angle (on page 794)
- M. To align a single angle to its flanges (on page 795)
- G.4.3 Relationship Between Global and Local Coordinates (on page 2301)
- M. To assign a member to a reference point (on page 796)
- M. To assign material constants (on page 793)
- G.17.3.3.1 Composite Damping (on page 2367)
- M. To assign a composite damping ratio (on page 874)
- G.17.3.3.1 Composite Damping (on page 2367)
- G.4.3 Relationship Between Global and Local Coordinates (on page 2301)
- G.6.8 Curved Members (on page 2329)
- G.4.3 Relationship Between Global and Local Coordinates (on page 2301)

TR.26.3 Surface Constants Specification

Explained below is the command syntax for specifying constants for surface entities.
Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

General Format

SURFACE CONSTANTS

\{ E f1 \ | \ G f2 \ | \ POISSON f3 \ | \ DENSITY f4 \ | \ ALPHA f5 \} \{ LIST surface-list \ | \ ALL \}

Where:

- **E f1**: Young's Modulus (E)
- **G f2**: Modulus of Rigidity (G)
- **POISSON f3**: Poisson's ratio
- **DENSITY f4**: weight density
- **ALPHA f5**: coefficient of thermal expansion

In lieu of numerical values, built-in material names may be used for the above specification of constants. The built-in names are STEEL, CONCRETE, and ALUMINUM. See TR.26.2 Specifying Constants for Members and Elements (on page 2503) for the values used in the built-in materials.

Example

SURFACE CONSTANTS
E 3150 LIST 1 TO 4
POISSON 0.17 ALL
DENSITY 8.68e-005 LIST 1 TO 4
ALPHA 5.5e-006 LIST 1 TO 4

Alternatively, where the material constants have been defined in a material object such as CONCRETE:

SURFACE CONSTANTS
MATERIAL CONCRETE LIST 1 TO 4

Notes

- **a.** If G is not specified, but Poisson is specified, G is calculated from \(E / (2 (1 + \text{Poisson}))\).
- **b.** If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
- **c.** If G and Poisson are both specified, the input value of G is used, G is not calculated in this situation.
- **d.** If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated from \(((E/2G) – 1)\).
- **e.** To obtain a report of the values of these terms used in the analysis, specify the command PRINT MATERIAL PROPERTIES.

Related Links

- **G.5.3 Surface Elements (Deprecated)** (on page 2322)
TR.26.4 Modal Damping Information

To define unique modal damping ratios for every mode, STAAD.Pro allows you to specify modal damping either directly or by using Rayleigh damping (the algebraic combination of mass-proportional damping and stiffness-proportional damping). This will be used in a dynamic analysis when MDAMP has been specified.

**Note:** If all modes have the same damping, then you may simply enter damping within the define time history load or within the dynamic loading commands.

Damping may be entered:

- by specifying that the program EVALUATE each mode’s damping based on the frequency of the mode and the minimum and maximum damping entered here. The formula used to evaluate the damping is given in Evaluate Damping (on page 2370).
- by specifying that the program CALCULATE each mode’s damping based on the frequency of the mode and the mass factor, ALPHA, and the stiffness factor, BETA. The formula used to calculate the damping is given in Calculate Damping (on page 2368).
- explicitly for some or all modes (EXPLICIT).

The damping entered will be used in time history load cases and in response spectrum load cases that use the CQC or ASCE4 methods and/or Spectra vs. Period curves versus damping.

**General Format**

```
DEFINE DAMPING INFORMATION
{ EVALUATE dmin (dmax) | CALCULATE ALPHA c1 BETA c2 (MAX dmax MIN dmin) | EXPLICIT d1 (d2 d3 d4 ... dn ) }
END
```

Where:

- `dmin` the minimum damping ratio to be used in the evaluate damping formula
- `dmax` the optional maximum damping ratio to be used in the evaluate damping formula
- `ALPHA c1` the mass-proportional damping coefficient, α, used in the calculate damping formula
- `BETA c2` the stiffness-proportional damping coefficient, β, used in the calculate damping formula
- `MAX dmax, MIN dmin` optional minimum and maximum damping ratios, respectively, used in the calculate damping formula
- `d1 d2 ... dn` the damping ratios for each mode given in the explicit method

**Note:** Damping ratios must be in the range 0.0 through 1.0.

**Related Links**

- [G.17.3.3.2 Modal Damping](on page 2368)
- [M. To explicitly define damping values for modes](on page 875)
- [M. To evaluate damping for modes](on page 875)
- [G.17.3.3 Damping Modeling](on page 2366)
- [G.17.3.3 Damping Modeling](on page 2366)
TR.26.5 Composite Damping for Springs

This command may be used to designate certain support springs as contributing to the computation of modal damping by the composite damping method. The Response Spectrum or Time History dynamic response analyses must select composite damping, CDAMP, for this data to have any effect on results and the material used must have suitable damping values, DAMP, specified.

General Format

```
SPRING DAMPING
```

```
joint-list *{ KFX f1 | KFY f2 | KFZ f3 }
```

Where:

- \( f_1, f_2, f_3 \) damping ratios (0.001 to 0.990) in the local X, Y, and Z directions, respectively

Note: At least one of KFX, KFY, or KFZ must be entered and each one entered must have a spring damp value following it.

Description

If this Spring Damping command is entered, then all springs in the structure are included in the composite damping calculation, otherwise no spring is considered in that calculation.

This input command does not create a spring. Rather, if a support spring exists at the joint in the specified direction, then it will be assigned the damping ratio.

Note: This is not a discrete damper definition.

TR.26.6 Member Imperfection Information

To define camber and drift specifications for selected members. Drift is usually for columns and camber for beams.
General Format

**DEFINE IMPERFECTION**

| CAMBER \{ Y | Z \} (f₁) RESPECT (f₂) | XR f₄ f₅ | YR f₄ f₅ | ZR f₄ f₅ | MEM memb-list | LIST memb-list | ALL |
|---------------------------------|---------|---------|---------|-------------|--------------|------|
| DRIFT \{ Y | Z \} (f₃)          | XR f₄ f₅ | YR f₄ f₅ | ZR f₄ f₅ | MEM memb-list | LIST memb-list | ALL |

- **f₁** Camber value. Default = 300
- **f₂** Respect value. Default = 1.6
- **f₃** Drift value. Default = 200.
- **f₄, f₅** global coordinate values to specify X, Y, or Z range for member selection.

Imperfections will be simulated by loads. These loads will be generated for the specified members if there is an Imperfection Analysis specified and if the specified members are active, in compression, and are not truss or tension/compression only members.

**Notes**

Camber is the maximum offset of the neutral axes in the defined direction from a vector that passes through the ends of the beam (i.e., the local X axis) defined as the ratio of member length/offset.

Drift is the offset of the end a member from its specified location defined as a ratio of member length/offset.
RESPECT is a non dimensional constant used to skip the camber imperfection calculation if the compressive load is small or EI is great or length is short. A combination of these terms is calculated and called EPSILON. If EPSILON is less than RESPECT, then the imperfection calculation is skipped for that local direction, for that case, for that member. The imperfection calculation is also skipped for any member that is in tension.

$$\text{EPSILON}_y = \text{Length} \times \sqrt{\frac{\text{abs(axial load)}}{\text{EI}_z}}$$

$$\text{EPSILON}_z = \text{Length} \times \sqrt{\frac{\text{abs(axial load)}}{\text{EI}_y}}$$

Example

```plaintext
SUPPORTS
1 FIXED
2 FIXED BUT FX
DEFINE IMPERFECTION
CAMBER Y 300 RESPECT 0.4 ALL
LOAD 1 GRAVITY + LATERAL
MEMBER LOAD
1 UNI GY -1
JOINT LOAD
2 FX -10
PERFORM IMPERFECTION ANALYSIS PRINT STATICS CHECK
```

Related Links

- M. To assign member imperfection for members (on page 800)
- M. To assign member imperfection for members (on page 800)
- G.17.2.4 Imperfection Analysis (on page 2355)

TR.27 Support Specifications

STAAD support specifications may be either parallel or inclined to the global axes.

See TR.27.1 Global Support Specification (on page 2514) for specification of supports parallel to the global axes.
See TR.27.2 Inclined Support Specification (on page 2516) for specification of inclined supports.

Related Links
TR.27.1 Global Support Specification

This set of commands may be used to specify the SUPPORT conditions for supports parallel to the global axes.

For SURFACE elements, if nodes located along a straight line are all supported identically, as in the case of the base of a wall, support generation can be performed for assigning restraints to those nodes. See the GENERATE option in the command syntax below. You only need to provide the starting and ending nodes of the range, and the type of restraint.

General Format

```
SUPPORTS
{ joint-list | ni TO nj GENERATE } { PINNED | FIXED ( BUT release-spec (spring-spec) ) | ENFORCED ( BUT release-spec ) }
release-sepc = { FX | FY | FZ | MX | MY | MZ }
弹簧-spec = *{ KFX f1 | KFY f2 | KFZ f3 | KMX f4 | KMY f5 | KMZ f6 }
```

Where:

- `ni, nj` = Start and end node numbers, respectively, for generating supports along a SURFACE element edge.
- `f_1 ... f_6` = Spring constants corresponding to support spring directions X, Y, and Z and rotations about X, Y, and Z, respectively.

Description of Pinned and Fixed

A PINNED support is a support that has translational, but no rotational restraints. In other words, the support has no moment carrying capacity. A FIXED support has both translational and rotational restraints. A FIXED BUT support can be released in the global directions as described in release-spec (FX for force-X through MZ for moment-Z). Also, a FIXED BUT support can have spring constants as described in spring-spec (translational spring in global X-axis as KFX through rotational spring in global Z-axis as KMZ). Corresponding spring constants are `f_1` through `f_6`. The rotational spring constants are always per degree of rotation. All six releases may be provided as may be required when using the CHANGE command. If both release specifications and spring specifications are to be supplied for the same support joint, release specifications must come first.

1. See TR.38 Change Specification (on page 2835) for information on specification of SUPPORTS along with the CHANGE command specifications.
2. Spring constants must be provided in the current units.
3. All spring DOF must be entered after the last non-spring DOF is specified, if both are on the same line.
4. If there are two entries for the same joint, then:
   a. any direction that is pinned/fixed on either will be fixed in that direction.
   b. any direction released on one and is a spring on the other will use the spring.
   c. any direction that is pinned/fixed on one and a spring on the other will use the spring.
   d. any direction that is a spring on two or more entries will have the spring constants added.
Example 1

SUPPORTS
1 TO 4 7 PINNED
5 6 FIXED BUT FX MZ
8 9 FIXED BUT MZ KFX 50.0 KFY 75.
18 21 FIXED
27 FIXED BUT KFY 125.0

In this example, joints 1 to 4 and joint 7 are pinned. No moments are carried by those supports. Joints 5 and 6 are fixed for all DOF except in force-X and moment-Z. Joints 8 and 9 are fixed for all DOF except moment-Z and have springs in the global X and Y directions with corresponding spring constants of 50 and 75 units respectively. Joints 18 and 21 are fixed for all translational and rotational degrees of freedom. At joint 27, all the DOF are fixed except the FY DOF where it has a spring with 125 units spring constant.

Description of Enforced Support

Enforced Support defines which translational and rotational directions, at a joint, may have a support displacement imposed. The support displacements are defined for each load case in TR.32.8 Support Joint Displacement Specification (on page 2683). If no support displacement is entered, then zero displacement will be imposed, as if that direction was FIXED. The enforced displacement directions will be fixed for dynamic load cases.

If there are two entries for the same joint, then any direction that is enforced on either will be enforced in that direction, overriding any other support specification for that joint-direction.

Currently the support generation command can only be used in conjunction with the Surface element support specifications.

Example 2

SUPPORTS
3 TO 7 GENERATE PIN

The above command will generate pinned supports for all joints located between nodes No. 3 and 7 along a straight line. This may include joints explicitly defined by the user or joints generated by the program for internal use only (e.g., as a result of SET DIVISION and SURFACE INCIDENCES commands).

Related Links

- M. To assign a fixed or pinned support (on page 812)
- G.13 Supports (on page 2335)
- M. To assign an enforced support (on page 813)
- G.13 Supports (on page 2335)
- EX. US-24 Analysis of a Concrete Block Using Solid Elements (on page 4552)
- EX. UK-24 Analysis of a Concrete Block Using Solid Elements (on page 4833)
- M. To assign a spring support (on page 814)
- G.13 Supports (on page 2335)
- EX. US-3 Soil Springs for Portal Frame (on page 4383)
TR.27.2 Inclined Support Specification

These commands may be used to specify supports that are inclined with respect to the global axes.

**General Format**

```
SUPPORT

joint-list INCLINED \{ f1 f2 f3 | REF f4 f5 f6 | REFJT f7 \} \{ PINNED | FIXED \ BUT release-spec \( spring-spec \) \} | ENFORCED \ BUT release-spec \}
```

Where:

- \( f1, f2, f3 \) \( x, y, z \) relative distances from the joint in the global directions to the reference point
- \( f4, f5, f6 \) \( x, y, z \) global coordinates of an arbitrary reference point
- \( f7 \) a joint number whose \( x, y, z \) global coordinates are the reference point

A vector from the joint location to the reference point location serves to define a local coordinate system (same as member with \( BETA = 0 \)). The inclined support directions are in this local "Inclined Support Axis System" (see more below).

**Note:** The release-spec and spring-spec are the same as in the Global Supports (See TR.27.1 Global Support Specification (on page 2514)). However, FX through MZ and KFX through KMZ refer to forces/moments and spring constants in the "Inclined Support Axis System" (see below).

**Inclined Support Axis System**

The INCLINED support specification is based on the "Inclined Support axis system." The local x-axis of this system is defined by assuming the inclined support joint as the origin and joining it with a "reference point" with coordinates of \( f1, f2, \) and \( f3 \) (see the following figure) measured from the inclined support joint in the global coordinate system.
The Y and Z axes of the inclined support axis system have the same orientation as the local Y and Z axes of an imaginary member whose BETA angle is zero and whose incidences are defined from the inclined support joint to the reference point. [G.4.3 Relationship Between Global and Local Coordinates](on page 2301) for more information on these concepts.

**Note:** Inclined support directions are assumed to be same as global when computing some dynamic and UBC intermediate results (e.g., global participation factors). If masses and/or forces in the free directions at inclined supports are a relatively small portion of the overall forces, then the effect should be very small.

**Example**

```
SUPPORT
4 INCLINED 1.0 -1.0 0.0 FIXED BUT FY MX MY MZ
```

**Related Links**

- [M. To assign an inclined support](on page 817)
- [G.13 Supports](on page 2335)
- [EX. US-19 Inclined Supports](on page 4515)
- [EX. UK-19 Inclined Supports](on page 4795)
- [G.13 Supports](on page 2335)

**TR.27.3 Automatic Spring Support Generator for Foundations**

STAAD.Pro has a facility for automatic generation of spring supports to model footings and foundation mats. This command is specified following the SUPPORT command.

**General Format**

```
SUPPORT
```
then either

\[
\{ \text{joint-list ELASTIC FOOTING } f_1 (f_2) \mid \text{joint-list ELASTIC MAT} \mid \text{plate-list PLATE MAT} \} \text{ DIR } \{ X \mid XONLY \mid Y \mid YONLY \mid Z \mid ZONLY \} \text{ SUBGRADE } f_3
\]

(\text{PRINT}) ( \{\text{COMP} \mid \text{MULTI} \} )

or

\[
\text{plate-list PLATE MAT DIR ALL SUBGRADE } f_4 (f_5 f_6)
\]

(\text{PRINT}) ( \{\text{COMP} \mid \text{MULTI} \} )

Where:

- **FOOTING** \( f_1 f_2 \) Length and width of the ELASTIC footing. If \( f_2 \) is not given, the footing is assumed to be a square with sides \( f_1 \).
- **X,Y,Z** Global direction in which soil springs are to be generated.
- **SUBGRADE** \( f_3 \) Soil subgrade modulus in current force/area/length units for elastic footings.
- **SUBGRADE** \( f_4 f_5 f_6 \) Soil subgrade modulus for mat foundations for use with ALL option in current force/area/length units in Y, X, Z directions respectively. \( f_4, f_5 \) default to \( f_3 \) if omitted.

**Note:** For mat foundation subgrade values, the Y value is specified first, followed by X and then Z.

![Correct and Incorrect order for specifying subgrade modulus values](image)

**Caution:** Do not use this command with SET Z UP.

**Description**

If you want to specify the influence area of a joint yourself and have STAAD simply multiply the area you specified by the sub-grade modulus, use the ELASTIC FOOTING option. Situations where this may be appropriate are such as when a spread footing is located beneath a joint where you want to specify a spring support. A value for \( f_1 \) (and \( f_2 \) if its a non-square footing) is required for the FOOTING option.

If you want to have STAAD calculate the influence area for the joint (instead of you specifying an area yourself) and use that area along with the sub-grade modulus to determine the spring stiffness value, use the ELASTIC MAT option. Situations where this may be appropriate are such as when a slab is on soil and carries the weight of the structure above. You may have modeled the entire slab as finite elements and wish to generate spring supports at the nodes of the elements.
The PLATE MAT option is similar to the Elastic Mat except for the method used to compute the influence area for the joints. If your mat consists of plate elements and all of the influence areas are incorporated in the plate areas, then this option is preferable. Enter a list of plates or YRANGE f1 f2 at the beginning of the command, the joint influence areas are then calculated using the same principles as joint forces would be from uniform pressure on these plates. This method overcomes a major limitation of the Delaunay triangle method used in the ELASTIC MAT option, which is that the contour formed by the nodes of the mat must form a convex hull.

The PLATE MAT DIR ALL option is similar to the Plate Mat except that the spring supports are in 3 directions. If the compression only option is also specified, then the compression direction will be assumed to be in the Y direction. If the Y spring at a joint goes slack (lift off), then the X and Z spring stiffnesses for that joint will also be set to zero. Otherwise the X and Z springs act in both directions. The influence area for the X and Z springs is the same as used for the Y spring. Three values of subgrade reaction may be entered, the first is for the Y direction, the second for X and the third for Z.

The keyword DIR is followed by one of the alphabets X, Y or Z (or XONLY, YONLY, or ZONLY) which indicate the direction of resistance of the spring supports. If X or Y or Z is selected then a spring support is generated in that direction plus 3 other directions receive a fixed support, e.g., if Y is selected, then FY is supported by a spring; FX and FZ and MY are fixed supports; and MX and MZ are free. If XONLY, YONLY, or ZONLY are selected then only a spring support in that direction is generated.

The keyword SUBGRADE is followed by the value of the subgrade reaction. The value should be provided in the current unit system signified by the most recent UNIT statement prior to the SUPPORT command.

The PRINT option prints the influence area of each joint.

Use the COMP option generated will be compression only springs.

Use the MULTI option to generate multilinear springs. Add the associated multilinear curve input after each MAT command (with the multi option) to describe the displacement-spring constant curve. See TR.27.4 Multilinear Spring Support Specification (on page 2520) for additional information on this input format. The actual spring constant used will be the subgrade modulus (f3 entered above) times the influence area (computed by STAAD) times the si values entered in the curve (so the curve stiffness values will likely be between 0.0 and 1.0).

SPRINGS d1 f1 d2 f2 … dn fn

Example

SUPPORTS
1 TO 126 ELASTIC MAT DIREC Y SUBG 200.
1 TO 100 PLATE MAT DIREC Y SUBG 200.
YR -.01 .01 PLA MAT DIR Y SUBG 200 MUL
SPRINGS -100 2.0 -0.51 2.0 -0.50 1.0 0.0 0.0 1000 0.0

The first command above instructs STAAD to internally generate supports for all nodes 1 through 126 with elastic springs. STAAD.Pro first calculates the influence area perpendicular to the global Y axis of each node and then multiplies the corresponding influence area by the soil subgrade modulus of 200.0 to calculate the spring constant to be applied to the node. In the 2nd example, the nodes of plates 1 to 100 are assigned spring supports, generated using a subgrade modulus of 200 units.

Notes

a. A closed surface is generated by the program based on the joint-list that accompanies the ELASTIC MAT command. The area within this closed surface is determined and the share of this area for each node in the list is then calculated.
Hence, while specifying the joint-list, one should make sure that these joints make up a closed surface. Without a proper closed surface, the area calculated for the region may be indeterminate and the spring constant values may be erroneous. Consequently, the list should have at a minimum, 3 nodes.

b. The internal angle formed by 2 adjacent segments connecting 3 consecutive nodes in the list should be less than 180 degrees. In other words, the region should have the shape of a convex polygon. The example below explains the method that may be used to get around a situation where a convex polygon is not available.

c. For the model comprised of plate elements 100 to 102 in the figure below, one wishes to generate the spring supports at nodes 1 to 8. However, a single ELASTIC MAT command will not suffice because the internal angle between the edges 1-8 and 8-7 at node 8 is 270 degrees, which violates the requirements of a convex polygon.

So, you should break it up into two commands:

\[
\begin{align*}
1 & \ 2 \ 3 \ 8 \ \text{ELASTIC MAT DIREC Y SUBG 200.} \\
3 & \ 4 \ 5 \ 6 \ 7 \ 8 \ \text{ELASTIC MAT DIREC Y SUBG 200.}
\end{align*}
\]

Joints 3 and 8 will hence get the contribution from both of the above commands.

The command works only when the plane of the closed region is parallel to one of the global planes X-Y, Y-Z or X-Z. For regions that are inclined to one of the global planes, the spring constant will have to be evaluated manually and specified using the FIXED BUT type of spring support.

**Related Links**

- M. To assign a foundation support (on page 818)
- G.13 Supports (on page 2335)
- EX. US-23 Spring Support Generation for a Slab on Grade (on page 4541)
- EX. UK-23 Spring Support Generation for a Slab on Grade (on page 4822)

**TR.27.4 Multilinear Spring Support Specification**

When soil is modeled as spring supports, the varying resistance it offers to external loads can be modeled using this facility, such as when its behavior in tension differs from its behavior in compression.
General Format

```
MULTILINEAR SPRINGS
```

```
joint-list SPRINGS d_1 s_1 d_2 s_2 ... d_n s_n
```

Where:

\[ d_i s_i \] pairs represent displacement and spring constant pairs \((s_i \) is zero or positive), starting from the maximum negative displacement to the maximum positive displacement.

The first pair defines the spring constant from negative infinity displacement up to the displacement defined in the second pair. The second pair define the spring constant when the support displaces in the range from the displacement defined in the second pair, up to the displacement defined in the third pair. This continues for each displacement and spring constant pair until the last pair which defines the spring constant for displacements greater than the displacement in the last pair to positive infinity.

Each load case in a multilinear analysis must be separated by a `CHANGE` command and have its own `PERFORM ANALYSIS` command. There may not be any `PDELTA`, `NONLIN`, dynamics, `CABLE`, or `TENSION/COMPRESSION` analysis included. The multilinear spring command will initiate an iterative analysis and convergence check cycle. The cycles will continue until the root mean square (RMS) of the effective spring rates used remain virtually the same for two consecutive cycles.

### Example

```
UNIT ...
SUPPORT
1 PINNED; 2 4 FIXED BUT KFY 40.0
MULTILINEAR SPRINGS
2 4 SPRINGS -1 40.0 -0.50 50.0 0.5 65.0
```

Load-Displacement characteristics of soil can be represented by a multilinear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. A typical spring characteristic of soil may be represented as the step curve as shown in the figure below. In the above example, the multilinear spring command specifies soil spring at joints 2 and 4. (Note that the amplitude of the step curve does not change after the first point.)

### Notes

a. `SUPPORT` springs must have previously been entered for each spring entered here. For the first cycle, the spring value used will be the support spring value (not the zero displacement value here). Use a realistic and stable value.

b. All directions that have been defined with an initial spring stiffness in the `SUPPORT` command will become multilinear with this one curve.

c. This command can be continued to up to 11 lines by ending all but last with a hyphen. The semi-colons and the `XRANGE`, `YRANGE`, `ZRANGE` list items may not be used.

d. This command needs a minimum of two displacement and spring constant pairs.
Spring Constant

-0.5 L 0.5 L

40 F/L

50 F/L

65 F/L

Displacement

Figure 299: Spring constant is always positive or zero. \( F = \text{Force Units}, L = \text{Length Units} \)

Multilinear springs should not be used in the following conditions:

- Modal dynamics
- Buckling analysis
- Imperfection analysis
- PDELTA analysis
- NONLINEAR analysis
- Advanced cable analysis
- Direct analysis
- Models with tension/compression members and/or supports
- Models with inclined supports

Related Links
- G.17.2.5 Multilinear Analysis (on page 2356)

TR.27.5 Spring Tension/Compression Specification

This command may be used to designate certain support springs as Tension-only or Compression-only springs.

General Format

<table>
<thead>
<tr>
<th>SPRING TENSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>joint-List (spring-spec )</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPRING COMPRESSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>joint-List (spring-spec )</td>
</tr>
</tbody>
</table>

where:

| spring-spec = *{ KFX | KFY | KFZ | ALL } |

STAAD.Pro 2522 User Manual
Description

Tension-only springs are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them. Compression-only springs are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them.

If no spring spec is entered or ALL is entered then all translational springs at that joint will be tension (or compression) only. This input command does not create a spring, only that if a support spring exists at the joint in the specified direction then it will also be tension (or compression) only. Refer to TR.27 Support Specifications (on page 2513) to define springs.

For compression only springs the ALL parameter has special meaning. The compression only spring is in the Y direction; the X and Z direction springs are bi-directional. However when the Y direction spring goes slack, the X and Z springs at the same joint are made inactive as well.

**Note:** The procedure for analysis of Tension-only or Compression-only springs requires iterations for every load case and therefore may be quite involved.

Since this command does not specify whether the spring is in the positive or negative direction from the joint, it is assumed in STAAD.Pro to be in the negative direction. For negative displacement the spring is in compression and for positive the spring is in tension.

If a CHANGE command is used (because of a change in list of tension springs, supports, etc.), then the SET NL command must be used to convey to STAAD.Pro that multiple analyses and multiple structural conditions are involved.

1. Refer to TR.5 Set Command Specification (on page 2413) for explanation of the SET NL command. The number that follows this command is an upper bound on the total number of primary load cases in the file.
2. STAAD.Pro performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a SET ITERLIM i command (with i > 10) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high. If not converged, a message will be in the output.
3. The principle used in the analysis is the following.
   - The program reads the list of springs declared as SPRING TENSION and/or COMPRESSION.
   - The analysis is performed for the entire structure and the spring forces are computed.
   - For the springs declared as SPRING TENSION / COMPRESSION, the program checks the axial force to determine whether it is tensile or compressive. Positive displacement is TENSION. If the spring cannot take that load, the spring is "switched off" from the structure.
   - The analysis is performed again without the switched off springs.
   - Up to ITERLIM iterations of the above steps are made for each load case.
   - This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable. You may need to include some support in each global direction that is not tension (or compression) only to be stable on every iteration.
4. A revised SPRING TENSION / COMPRESSION command and its accompanying list of joints may be provided after a CHANGE command. If entered, the new SPRING commands replace all prior SPRING commands. If not entered after a CHANGE, then the previous spring definitions are used.
5. The SPRING TENSION command should not be used if the following load cases are present: Response Spectrum load case, Time History Load case, Moving Load case. If used, the SPRING TENSION / COMPRESSION will be ignored in all load cases.
6. If the SPRING TENSION / COMPRESSION command is used in a model with UBC, IBC or other such seismic load cases, each such load case must be followed by an ANALYSIS and CHANGE command.
**Notes**

a. A spring declared as tension-only or compression-only will carry axial forces only. It will not carry moments.
b. The SPRING TENSION / COMPRESSION commands should not be specified if the INACTIVE MEMBER command is specified.
c. Do not use Load Combination to combine these cases. Tension/Compression cases are non-linear and should not be linearly combined as in Load Combination. Use a primary load case with the Repeat Load command.

**Example**

```
SPRING TENSION
12 17 19 TO 37 65
SPRING COMPRESSION
5 13 46 TO 53 87 KFY
```

The following is the general sequence of commands in the input file if the SPRING TENSION or COMPRESSION command is used. This example is for the SPRING TENSION command. Similar rules are applicable for the SPRING COMPRESSION command. The dots indicate other input data items.

```
STAAD ...
SET NL ...
UNITS ...
JOINT COORDINATES ...
MEMBER INCIDENCES ...
ELEMENT INCIDENCES ...
CONSTANTS ...
MEMBER PROPERTY ...
element property ...
SUPPORTS ...
spring TENSION ...
LOAD 1 ...
LOAD 2 ...
LOAD 3
REPEAT LOAD ...
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ...
PRINT ...
PARAMETER ...
CHECK CODE ...
FINISH
```
TR.28 Rigid Diaphragm Modeling

STAAD.Pro has two methods for defining rigid floor diaphragms. Both are functionally equivalent but the Master-Slave joint requires that an analytical node be located and specified as the master node. For most models, the simpler Floor Diaphragm option is preferred.

TR.28.1 Master/Slave Specification

This set of commands may be used to model specialized linkages (displacement tying, rigid links) through the specification of master and slave joints. Please read the notes for restrictions.

General Format

```
SLAVE *{ XY | YZ | ZX | RIGID | FX | FY | FZ | MX | MY | MZ } MASTER j JOINT joint-spec
```

where:

```
joint-spec = { joint-list | *( XRANGE | YRANGE | ZRANGE } f1 f2 }
```

Description

The master/slave option provided in STAAD allows the user to model specialized linkages (displacement tying, rigid links) in the system. For example, SLAVE FY ... connects the two joints such that the Y displacement at the slave will be the sum of Y displacement at the master plus the rigid rotation,

\[ R \sin \theta \]

Note that instead of providing a joint list for the slaved joints, a range of coordinate values (in global system) may be used. All joints whose coordinates are within the range are assumed to be slaved joints. For convenience, the coordinate range specified for slaved joints in this entry may include the master joint for this entry. However, master and slave joints of other entries must not be included in the coordinate range. All 2 or 3 ranges can be entered to form a “tube” or “box” for selecting joints in the tube or box region.

Fx, Fy etc. are the directions in which they are slaved (any combination may be entered).

If two or more entries have the same master, the slave lists will be merged. Please ensure that the same direction specs are used.

The direction specifiers (XY, YZ, ZX) are combinations of the basic directions, XY is the same as entering FX, FY, MZ, etc. Any combination of direction specifiers may be entered. An example of the use of this format is: a rigid diaphragm floor that still retains bending flexibility entered as SLA ZX ...

If RIGID or if all directions are provided, the joints are assumed to be rigidly connected as if SLA DIA RIG were entered, even if DIA is omitted.

**Note:** Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.
Restrictions

- Solid elements may not be connected to slave joints.
- Master joints may not be master or slaves in another entry.
- Slave joints may not be master or slaves in another entry.
- Slave directions at joints may not be supported directions or have displacements imposed.
- Master and/or slave joints may not be inclined supports.
- The master/slave specification is only intended for linear static and dynamic analysis.
- Multilinear springs are not permitted.

The internal processing of any master/slave command includes an automatic bandwidth reduction.

**Note:** In versions of STAAD.Pro prior to TAAD.Pro 2007 Build 1001, for the bandwidth reduction to take place, the data of master/slave would need to be repeated before the support definitions. This requirement is now no longer required.

```
Example

Fully Rigid and Rigid Floor Diaphragm

SLAVE RIGID MASTER 22 JOINT 10 TO 45
SLAVE RIGID MASTER 70 JOINT YRANGE 25.5 27.5
SLA ZX MAS 80 JOINT YR 34.5 35.5
```

Related Links

- [G.14 Rigid Diaphragms](on page 2336)
- [M. To assign a rigid link between nodes](on page 810)
- [G.14 Rigid Diaphragms](on page 2336)

**TR.28.2 Floor Diaphragm**

This command is used to create rigid floor diaphragms without the need to specify a master joint at each. When specified, this command directs the engine to perform the following:

a. calculate the center of mass for each rigid diaphragm (where master joint is to be located) considering the mass model of the structure. The mass must be modeled using mass reference load. See [TR.31.6 Defining Reference Load Types](on page 2642)

b. create, internally, an analytical node at the center of mass location to be included during analysis (unless a master node is specified) if an existing analytical node exists at this point, then the existing joint is used in lieu of creating a new joint.

**Tip:** The center of mass of each diaphragm is included in the preprocessing output. To include the center of rigidity in the post-processing output, you must include the PRINT DIAPHRAGM CR command. See [TR.42 Print Specifications](on page 2840)

c. search all nodes available within a diaphragm and add them as slave nodes; with the master node located at the center of mass for the diaphragm (or at the specified master node)
After all diaphragms are defined, the following optional commands can be added:

( CHECK SOFT STORY [ 1893 | 1893 2016 | ASCE7 ] )

( CHECK IRREGULARITIES CODE 1893 2016 )

where:

\[ \text{diaphragm-spec} = \text{HEIGHT } f_1 \ ( \text{MASTER } i_2 ) \ ( \text{JOINT } \text{joint-list} ) \]

or

\[ \text{diaphragm-spec} = \text{YRANGE } f_2 \ f_3 \ ( \text{MASTER } i_2 ) \ ( \text{JOINT XRANGE } f_4 \ f_5 \ ZRANGE \ f_6 \ f_7 ) \]

or

\[ \text{diaphragm-spec} = ( \text{MASTER } i_2 ) \ \text{JOINT } \text{joint-list} \]

**DIAPHRAGM i1**  
Diaphragm identification number

**BASE b1**  
base/ground floor level of the structure when not at the minimum Y coordinate defined in the model

**HEIGHT f1**  
Global coordinate value, in Y direction, to specify the floor level

**YRANGE f2 f3**  
Global coordinate values to specify a Y range, where \( f_2 \) is the lower bound and \( f_3 \) is the upper bound. The diaphragm is considered to be located at that floor height.

**XRANGE f4 f5**  
Global coordinate values to specify an X range. The diaphragm is considered to be located between this X range. If full floor is to be considered as only one diaphragm there is no need to define X range.

**ZRANGE f6 f7**  
Global coordinate values to specify Z range. The diaphragm is considered to be located between this Z range. If full floor is to be considered as only one diaphragm there is no need to define Z range.

**MASTER i2**  
User specified master joint number at the specified floor level. If not defined, the program will automatically calculate this joint as the diaphragm center of mass.

Instead of providing height or Y-range, joint lists can be provided to indicate the number of joints present at a particular floor level which will be connected to a master joint (either specified or calculated by the program).

**Soft Story and Irregularities Checks**

Refer to TR.28.2.1 Soft Story Checking (on page 2529) and TR.28.2.2 Check Irregularities (on page 2531) for details.
Notes

a. One full diaphragm definition should be provided per line. However, if there is joint-list, the list can extend to the second line with a continuation sign ("-".

<table>
<thead>
<tr>
<th>DIA</th>
<th>TYP</th>
<th>RIG</th>
<th>YR</th>
<th>JOI</th>
<th>XR</th>
<th>f1</th>
<th>f2</th>
<th>f3</th>
<th>f4</th>
<th>f5</th>
<th>f6</th>
<th>f7</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIA</td>
<td>f1</td>
<td>TYP</td>
<td>RIG</td>
<td>YR</td>
<td>f2</td>
<td>f3</td>
<td>JOI</td>
<td>XR</td>
<td>f4</td>
<td>f5</td>
<td>ZR</td>
<td>f6</td>
</tr>
<tr>
<td>DIA</td>
<td>f11</td>
<td>TYP</td>
<td>RIG</td>
<td>YR</td>
<td>f21</td>
<td>f31</td>
<td>JOI</td>
<td>XR</td>
<td>f41</td>
<td>f51</td>
<td>ZR</td>
<td>f61</td>
</tr>
<tr>
<td>DIA</td>
<td>f12</td>
<td>TYP</td>
<td>RIG</td>
<td>JOI</td>
<td>35</td>
<td>45</td>
<td>51</td>
<td>TO</td>
<td>57</td>
<td>59</td>
<td>TO</td>
<td>83</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>TO</td>
<td>90</td>
<td>TO</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

where \( f_1, f_{11}, \) and \( f_{12} \) are three rigid diaphragms located at floor height ranging between \( f_2 \) and \( f_3 \), \( f_{21} \) and \( f_{31} \), and the joints lying in the plane as indicated by their global Y coordinates respectively.

b. Diaphragms should be specified in ascending order (i.e., diaphragms at first floor level should be specified first before specifying that on 2nd floor level and so on).

c. If a user-defined master joint is specified in one diaphragm, then user-defined master joints should be specified for all diaphragms. Combination of user-defined master joint for one diaphragm and program calculated master joint for another diaphragm is not supported.

d. The mass model (in terms of reference load) must be specified before specifying floor diaphragm.

e. Floor diaphragms can be specified only once in an input file.

f. Floor diaphragm cannot be specified along with the FLOOR HEIGHT command (refer to TR.31.2.5.2 Identification of Floor Level (on page 2581)).

g. Floor diaphragm cannot be specified along with the MASTER-SLAVE command.

h. Floor diaphragm cannot be specified with the SET Z UP command.

i. Sloped diaphragms are not supported.

j. Base level (or ground floor level or support level) is taken as the minimum of Y coordinates defined. Different base level can be specified using the BASE \( b_1 \) option in the command. If used, this option must be the last line of the floor diaphragm system.

k. The maximum number of diaphragms allowed by the program (default value) is 150. If more than 150 diaphragms need to be specified, then SET RIGID DIAPHRAGM \( n \) must be specified before specifying joint incidence, where \( n \) = total number of diaphragms in the structure.

Example

*****************************************************************
UNIT FEET KIP
DEFINE REF LOAD
LOAD R1 LOADTYPE MASS
  * MASS MODEL
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FX 2.5 FY 2.5 FZ 2.5
49 TO 64 FX 1.25 FY 1.25 FZ 1.25

* LOAD R2
SELFWEIGHT Y -1
JOINT LOAD
17 TO 48 FY -2.5
49 TO 64 FY -1.25
END DEF REF LOAD
******************************************************************
Related Links

- A. To check for soft stories and seismic code irregularities (on page 937)
- TR.31.2.14 IBC 2015 Seismic Load Definition
- A. To check for soft stories and seismic code irregularities (on page 937)
- TR.31.2.14 IBC 2015 Seismic Load Definition (on page 2600)
- M. To assign nodes to a floor diaphragm (on page 811)
- A. To check for soft stories and seismic code irregularities (on page 937)
- G.17.3.2 Mass Modeling (on page 2365)

**TR.28.2.1 Soft Story Checking**

STAAD.Pro will perform this check when the option CHECK SOFT STORY command is used following a floor diaphragm. If omitted, no soft story check is performed. Soft story checking is only valid for structures having vertical elements in the form of columns and shear wall (without opening). When used, the program will exclude the effect of any other forms of lateral force resisting structural elements.

**General Format**

**Note:** This command is inserted after any pre-analysis PRINT commands.

```
CHECK SOFT STORY { 1893 | 1893 2016 | ASCE7 }
```

**Tip:** Include the PRINT STORY STIFFNESS command in the post-analysis print commands in order to review the story stiffness values.

**Description**

The option check for soft story can be performed per the IS 1893:2002, IS 1893-2016, or ASCE 7-95 codes.

A soft story building is a multi-story building where one or more floors are soft due to structural design. These floors can be dangerous in earthquakes, because they cannot cope with the lateral forces caused by the swaying of the building during a quake. As a result, the soft story may fail, causing what is known as a soft story collapse.

Soft story buildings are characterized by having a story which has a lot of open space. Parking garages, for example, are often soft stories, as are large retail spaces or floors with a lot of windows. While the unobstructed space of the soft story might be aesthetically or commercially desirable, it also means that there are less opportunities to install shear walls, specialized walls which are designed to distribute lateral forces so that a building can cope with the swaying characteristic of an earthquake.
If a building has a floor which is 70% less stiff than the floor above it, it is considered a soft story building. This soft story creates a major weak point in an earthquake, and since soft stories are mostly associated with retail spaces and parking garages, they are often on the lower stories of a building, which means that when they collapse, they can take the whole building down with them, causing serious structural damage which may render the structure totally unusable.

When used, the program checks for soft stories per Clause 7.1 Table 5 i)a and i)b for IS 1893 and per Clause 12.3.3 Table 12.3-2 1a and 1b for ASCE 7-05 (used for seismic design categories D, E, and F).

Stiffness Irregularities

- **Stiffness Irregularities: Soft Story** – As per this provision of the code, a soft story is one in which the lateral stiffness is less than 70 percent of that in the story above or less than 80 percent of the average lateral stiffness of the three story above.

- **Stiffness Irregularities: Extreme Soft Story** – As per this provision of the code, a extreme soft story is one in which the lateral stiffness is less than 60 percent of that in the story above or less than 70 percent of the average lateral stiffness of the three story above.

Thus, if any story of a building is found to be soft or extremely soft, the building is likely to suffer much damage in an earthquake than a similar type of building but has more regular vertical stiffness.

### Example

<table>
<thead>
<tr>
<th>FLOOR DIAPHRAGM</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIA 1 TYPE RIG HEI 3</td>
</tr>
<tr>
<td>DIA 2 TYPE RIG HEI 6</td>
</tr>
<tr>
<td>DIA 3 TYPE RIG HEI 9</td>
</tr>
<tr>
<td>CHECK SOFT STORY 1893 2016</td>
</tr>
</tbody>
</table>

### Example Output

**SOFT STORY CHECK**

<table>
<thead>
<tr>
<th>STORY</th>
<th>FL. LEVEL IN METE</th>
<th>S T A T U S</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.00</td>
<td>OK</td>
</tr>
<tr>
<td>2</td>
<td>6.00</td>
<td>OK</td>
</tr>
<tr>
<td>3</td>
<td>9.00</td>
<td>OK</td>
</tr>
</tbody>
</table>

**NOTE : NO SOFT STOREY IS DETECTED.**

### Related Links

- **A. To check for soft stories and seismic code irregularities** (on page 937)
- **TR.31.2.14 IBC 2015 Seismic Load Definition**
  - **A. To check for soft stories and seismic code irregularities** (on page 937)
  - **TR.31.2.14 IBC 2015 Seismic Load Definition** (on page 2600)
  - **A. To check for soft stories and seismic code irregularities** (on page 937)
TR.28.2.2 Check Irregularities

STAAD.Pro can check irregularities per the IS 1893 2016 seismic code.

Horizontal irregularities (torsional and reentrant corners) per Table 5 and vertical irregularities (mass irregularities and irregular modes of oscillation) per Table 6 of IS 1893 2016 can be checked by the program using this command.

**General Format**

<table>
<thead>
<tr>
<th>Note: This command is inserted after any pre-analysis PRINT commands.</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHECK IRREGULARITIES CODE IS1893 2016</td>
</tr>
</tbody>
</table>

| Note: Soft story checks per Table 6(i) can also be performed by the program. Refer to [TR.28.1 Soft Story Checking](on page 2529). |

**Torsion**

Torsion checks per Table 5(i)-a are performed. Unit loads in each orthogonal direction are applied to the master node of the diaphragm. The analysis is run and the displacements of the two extreme ends of the diaphragm are extracted from the analysis results. This ratio of these displacements is then checked against the code limit of 1.5.
The unit load, $F_x$, is applied at each diaphragm in the X-direction. After analysis, the program locates the extreme nodes of the diaphragm and then $d_{\text{min}}$ and $d_{\text{max}}$ are calculated. The ratio of $d_{\text{max}}/d_{\text{min}}$ is evaluated and compared to the code limit of 1.5. This process is then repeated in the Z direction for this diaphragm. Then the similar process is applied to the other diaphragms in the structure.

**Note:** Checks mandated by Table 5(i)-b are not implemented as the prerequisite modelling information to determine the torsional mode is not present in STAAD.Pro.

**Reentrant Corners**

The program will check for reentrant corners per Table 5(ii) of IS 1893 2016. The program first automatically identifies the boundary topology of the diaphragm. That is, the order in which analytical members and nodes of the boundary of the diaphragm are joined together to form a closed polygon. The program then determines the reentrant, or concave, corners of the diaphragm. The two analytical members joining the reentrant node is determined, their lengths are calculated, and then the projections of those lengths are calculated. To determine the ratios, the following are calculated:

$$L_i \times \cos(\alpha)/L_x \leq 0.15$$

$$L_i \times \cos(\alpha)/L_z \leq 0.15$$

If both members from the reentrant node are orthogonal, then for each member, only 1 ratio is calculated as $L_i/L_x$ for members oriented along the x-direction and $L_i/L_z$ for members oriented along the z-direction.
Figure 301: Example reentrant corner topology

**Note:** Cantilever members that do not form part a closed polygon are identified. These members are ignored by the program when evaluating reentrant corners.

All slave nodes that form part of a diaphragm must lie within the same plane. Further, the boundary slave nodes must form a closed polygon. Improper modeling of diaphragms may yield incorrect irregularity status results.

**Note:** Member offsets both in plan as well as in elevation may result in the reentrant corner check to be incorrect. You must exercise caution while using offsets and should check such joints manually.

### Mass Irregularities

The program will check for mass irregularities per Table 6(ii) of the IS 1893 2016 code. The mass of each floor is calculated as the total of all floor diaphragm masses at the same level. The ratio of each floor mass to the floor above and below is calculated. These ratios are compared to the code stipulated values.

### Irregular Modes of Oscillation

**Note:** The program will check for irregular modes of oscillation per the IS 1893 2016 code. This check is performed only if you have include the static seismic IS 1893 2016 definition using zone 4 or 5.

Per Table 6(vii)-a, the sum of the percentage of mass participation for the first 3 modes should be greater than 65%. This is checked in both orthogonal directions.
Per Table 6(vii)-b, the time period for the fundamental modes in one direction should differ by at least 10%.

**Example**

FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3  
DIA 2 TYPE RIG HEI 6  
DIA 3 TYPE RIG HEI 9  
CHECK IRREGULARITIES CODE IS1893 2016

**Example Output**

-IRREGULARITY CHECKS

STAAD.PRO IRREGULARITIES CHECK - ( IS1893-2016 ) v1.0
******************************************************************************

--TORSION IRREGULARITY CHECKS

Torsion Irregularity Check
Ref: Table 5 (i)  - Ratio Limit: 1.50

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Extreme Points of Dia in X</th>
<th>Extreme Points of Dia in Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-----</td>
<td>-----</td>
<td>-----</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0.04678</td>
</tr>
<tr>
<td>2</td>
<td>36</td>
<td>0.13804</td>
</tr>
</tbody>
</table>

Diaphragm DX-max/min DZ-max/min Status

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Extreme Points of Dia in X</th>
<th>Extreme Points of Dia in Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-----</td>
<td>-----</td>
<td>-----</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>1.0000</td>
</tr>
</tbody>
</table>

Diaphragm DX-max/min DZ-max/min Status

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Extreme Points of Dia in X</th>
<th>Extreme Points of Dia in Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>-----</td>
<td>-----</td>
<td>-----</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>1.0000</td>
</tr>
</tbody>
</table>

---GEOMETRY IRREGULARITY CHECKS

Re-Entrant Corner Check.  
(Ref: Table 5 (ii)  - Ratio Limit: 0.15 )

<table>
<thead>
<tr>
<th>Node</th>
<th>Re-Entrant X-Proj</th>
<th>X-Proj/Lx</th>
<th>Z-Proj</th>
<th>Z-Proj/Lz</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>5</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>8</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>4.0000</td>
</tr>
<tr>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>12</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>26</td>
<td>25</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>29</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>30</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>32</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
<tr>
<td>33</td>
<td>34</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>35</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
36-28 4.0000 0.3333 Re-Entrant
27 4.0000 0.3333 0.0000 0.0000

Diaphragm: Lx: Lz:
( m) ( m)
------------------------------
1 12.0000 12.0000
2 12.0000 12.0000

--MASS IRREGULARITY CHECKS

Mass Irregularity Check
Ref: Table 6 (ii) - Ratio Limit: 1.50

Dia. Level Mass Above Below Ratio Ratio Status
( m) ( kN) ( kN) ( kN) Above Below

---------------------------------------------------------------------
1 0.000 636.163 459.451 Base 1.385 N/A OK
2 5.000 459.451 Top 636.163 N/A 0.722 OK

***NOTE: Linear dynamic analysis needs to carried out for Irregular Modes of Oscillation check.

***NOTE: Static Seismic Loads for relevant code needs to be defined with Zone 4 and 5 for Irregular Modes of Oscillation check.

Related Links
- A. To check for soft stories and seismic code irregularities (on page 937)
- V. IS 1893 2016 GL Calculation (on page 3497)
- V. IS 1893 2016 Irregular Modes of Oscillation (on page 3509)
- V. IS 1893 2016 Mass Irregularity (on page 3519)
- V. IS 1893 2016 Re-entrant Corners (on page 3530)

TR.29 Definition of Member Attributes

Used to contain additional member attributes typically inserted from the STAAD.Pro User Interface.

General Format

Note: This section immediately follows MEMBER RELEASE data.

DEFINE MEMBER ATTRIBUTE

"attr-name " "attr-value " LIST member-list

...

END DEFINE

Note: The attribute name and attribute value are surrounded in quote mark (") characters.

Where
attr-name  The name of the attribute being used.
attr-value  The value of the current attribute. Values are specifically defined for each attribute name. Refer to the attribute types for explanations of appropriate values.
member-list  A list of member numbers.

Example

An example of some member attribute definitions in a STAAD.Pro input file:

```
DEFINE MEMBER ATTRIBUTE
"STRUCLINK" "BREAK_MEMBER_ACROSS" LIST 6 8
"CONTAG" "START:SHEAR:SS" LIST 4
END DEFINE
```

TR.29.1 Struclink Member Attribute

Used to provide identify continuity which analytical model members in the STAAD input file should be made continuous physical members across a joint passing data through the Struclink utility.

Caution: Though it is possible to manually enter Struclink data in the STAAD input file, it is strongly recommended that the user interface be used to add this information.

General Format

Note: A STRUCLINK attribute must be inserted in the DEFINE MEMBER ATTRIBUTE section.

```
"STRUCLINK" "{ USE_DEFAULT | BREAK_MEMBER_ACROSS }" LIST member-list
```

Where

**USE_DEFAULT**  This attribute value indicates that two adjacent members which meet the requirements for merging into a physical member will do so when exported via Struclink.

**BREAK_MEMBER_ACROSS**  This attribute value indicates that two adjacent members will not be merged into a single physical member when exported via Struclink, even if they would meet the criteria to do so. This allows you to define where physical members should be broken at a joint. For example, where two identical beams frame into either side of a multi-story column.

**member-list**  A list of member numbers.

Example

An example of some physical member formation attributes for Struclink in a STAAD.Pro input file:

```
DEFINE MEMBER ATTRIBUTE
"STRUCLINK" "USE_DEFAULT" LIST 1 TO 5 7
```
TR.29.2 Connection Tag Member Attribute

Used to assign members connection tag data from an associated Connection Tag XML file which is used for storing connection data and checking connection capacities.

**Caution:** Though it is possible to manually enter connection tag data in the STAAD input file, it is strongly recommended that the user interface be used to add this information as to ensure compatibility with the associated Connection Tags XML file. Refer to “Connection Tags sub menu” (on page 999)” in the STAAD User Interface help for additional information on using this feature.

**Note:** This feature is available in STAAD.Pro V8i (SELECTseries 4), release 20.07.09.22 and higher.

**General Format**

A CONTAG attribute must be inserted in the DEFINE MEMBER ATTRIBUTE section.

The attribute value for the CONTAG attribute is delimited into three parts: the end of the member, the tag category, and the tag name to use for this tag assignment. The later two parts are used to look up the connection tag releases and capacities in the Connection Tag XML file.

```
"CONTAG" "{ START | END }:tag-category :tag-name " LIST member-list
```

**Example**

An example of a pair of connection tags in a STAAD.Pro input file:

```
DEFINE MEMBER ATTRIBUTE
"CONTAG" "START:MOMENT:EM" LIST 6
"CONTAG" "START:SHEAR:SS" LIST 4
END DEFINE
```

The corresponding Connection Tag XML file would have —as a minimum— the following data:

```xml
<?xml version="1.0" encoding="utf-8" ?>
<ConnectionTagFile xmlns="http://tempuri.org/XMLSchema.xsd">
  <FileVersion value="1.0" />
  <Categories>
    <Category CategoryName="MOMENT">
      <CategoryDesc>End Moment Connection</CategoryDesc>
    </Category>
    <Category CategoryName="SHEAR">
      <CategoryDesc>Single Shear Connection</CategoryDesc>
    </Category>
  </Categories>
</ConnectionTagFile>
```
TR.30 Miscellaneous Settings for Dynamic Analysis

When dynamic analysis such as frequency and mode shape calculation, response spectrum analysis and time history analysis is performed, it involves eigenvalue extraction and usage of a certain number of modes during the analysis process. These operations are built around certain default values. This section explains the commands required to override those defaults.

Related Links
- G.17.3 Dynamic Analysis (on page 2362)
- G.17.3 Dynamic Analysis (on page 2362)
TR.30.1 Cut-Off Frequency, Mode Shapes, or Time

These commands are used in conjunction with dynamic analysis. They may be used to specify the highest frequency or the number of mode shapes that need to be considered.

General Format

CUT (OFF) { FREQUENCY f1 | MODE SHAPE i1 ( SHIFT MODE i2 ) (FREQ f2 ) | TIME t1 }

Where:

- **FREQUENCY f1**  Highest frequency (cycle/sec) to be considered for dynamic analysis.
- **MODE SHAPE i1**  Number of mode shapes to be considered for dynamic analysis. If the cut off frequency command is not provided, the cut off frequency will default to 108 cps. If the cut off mode shape command is not provided, the first six modes will be calculated. These commands should be provided prior to the loading specifications.
- **SHIFT MODE i2**  Number of eigen vectors per shift. This input may be used only when the solver fails to extract i1 number of modes due to insufficient memory.
- **FREQUENCY f2**  Frequency (in Hz) to be used as initial shift in Arnoldi/Lanczos method (i.e., it is not applicable to the basic solver). This is an optional input. If provided, the solver looks for eigenvalues close to the shift. The eigen values found may not necessarily be the smallest eigenvalues (i.e., closest to zero). If a full scale eigen solution is required, this value should not be defined. The engine default value in 0.0 (zero)
- **TIME t1**  Ending time for a time history analysis. If zero (default), the time history will end when the last forcing function ends.

A maximum of i1 mode shapes will be computed regard-less of f1. If during convergence testing, the 0 through f1 frequencies are converged, then the modal calculation will be completed before i1 mode shapes are calculated.

Basic Solver

If the CUT OFF FREQ f1 and CUT OFF MODE i1 commands are both entered, then after completing each iteration step in the subspace iteration, convergence testing is performed. If every frequency from 0.0 to f1 meets the convergence tolerance, then the subspace iteration is done. Similarly, if every mode from 0 to i1 meets the convergence tolerance, then the subspace iteration is done. If the cut off frequency f1 results in fewer modes than i1, then only those frequencies up to the cut off are used. If the cut off frequency would result in more modes than i1, then only the first i1 modes are used. That is, the modes cut off takes precedence over the frequency cut off.

Advanced Math Solver

A maximum of i1 mode shapes will be computed regard-less of f1. If the CUT OFF FREQ f1 and CUT OFF MODE i1 commands are both entered, the program will report only those modes that lie within f1 frequency.

Related Links

- M. To add a SNiP wind load definition (on page 837)
TR.30.2 Mode Selection

This command allows specification of a reduced set of active dynamic modes. All modes selected by this command remain selected until a new `MODE SELECT` is specified.

This command is used to limit the modes used in dynamic analysis to the modes listed in this command and deactivate all other modes that were calculated but not listed in this command. If this command is not entered, then all modes calculated are used in the dynamic analysis.

**General Format**

```
MODE SELECT mode-list
```

**Example**

```
CUT OFF MODES 10
MODE SELECT 1 TO 3
```

In this example, 10 modes will be calculated but only modes 1 and 3 will be used in dynamic analysis.

**Notes**

a. Do not enter this command within the loads data (from the first Load command in an analysis set down to the associated Analysis command).

b. The advantage of this command is that you may find the amount of structural response generated from a specific mode or a set of modes. For example, if 50 modes are extracted, but the effect of just the 40th to the 50th mode in a response spectrum analysis is to be determined, you may set the active modes to be 40 through 50. The results will then be devoid of any contribution from modes 1 through 39.

**Related Links**

- *G.17.3 Dynamic Analysis* (on page 2362)
Definition of the load systems must be provided before any primary load case is specified. This section describes the specification of load systems. Information on how to generate primary load cases using the defined load systems is available in TR.32.12 Generation of Loads (on page 2771).

**Note:** UBC loads do not fully consider the effects of forces at inclined support directions or at slave joint directions. Applying forces at these locations may introduce errors that are generally small.

**Related Links**
- G.16 Load Generator (on page 2343)

## TR.31.1 Definition of Moving Load System

This set of commands may be used to define the moving load system. Enter the DEFINE command only once with up to 100 TYPE commands.

The MOVING LOAD system may be defined in two possible ways: directly within the input file or using an external file.

### General Format

```
DEFINE MOVING LOAD (FILE file-name)
TYPE j { LOAD f1 f2 ... fn ( DISTANCE d1 d2 ... dn-1 (WIDTH w) ) | load-name (f) }
( DISTANCE d1 d2 ... dn-1 (WIDTH w) ) optionally as 2nd set
```

The FILE option should be used only in the second case when the data is to be read from an external file. The file name should be limited to 24 characters.

**Note:** Moving Loads can be generated for frame members only. They will not be generated for finite elements.

**Note:** All loads and distances are in current unit system.

### Define Moving Load Within the Input File

Use the first TYPE specification. Input Data must be all in one line (as shown above) or two sets of lines. If two sets, then the second set must begin with DIS as shown above. If two sets, then Load and Dist lines may end each line but last of each set with a hyphen (See example).

```
TYPE j LOAD f1 f2 ... fn
DISTANCE d1 d2 ... dn-1 (WIDTH w)
```

Where:

- **TYPE j** moving load system type number (integer limit of 200 types)
- **n** number of loads (eg, axles), 2 to 200.
- **LOAD fi** value of concentrated i\(^{th}\) load
- **DISTANCE di** distance between the (i+1)^{th} load and the i^{th} load in the direction of movement
- **WIDTH w** spacing between loads perpendicular to the direction of movement (e.g., the width of vehicle). If left out, one dimensional loading is assumed. This parameter will double the total load since the \(f_i\) is applied to each wheel.
Note: For a single moving load use: TYPE \_ \_ \_ j \_ \_ LOAD \_ \_ f1 \_ \_ DIST \_ \_ \_ 0

Define Moving Load Using an External File

Use the second TYPE specification.

\texttt{TYPE \_ \_ \_ j \_ \_ load-name \_ \_ (f)}

Where:

\begin{itemize}
  \item \texttt{TYPE j} \quad \text{moving load system type no. (integer).}
  \item \texttt{Load-name} \quad \text{the name of the moving load system (maximum of 24 characters).}
  \item \texttt{f} \quad \text{Optional multiplying factor to scale up or down the value of the loads. (default = 1.0)}
\end{itemize}

Following is a typical file containing the data.

\begin{verbatim}
CS200
50. 80. 90. 100.
7. 7. 9.
6.5
\end{verbatim}

\textbf{Figure 302: Graphical representation of the previous load system}

Several load systems may be repeated within the same file.

The STAAD moving load generator assumes:

1. All positive loads are acting in the negative global vertical (Y or Z) direction. The user is advised to set up the structure model accordingly.
2. Resultant direction of movement is determined from the X and Z (or Y if Z is up) increments of movements as provided by the user.

Reference Load

The first specified concentrated load in the moving load system is designated as the reference load. While generating subsequent primary load cases, the initial position of the load system and the direction of movement are defined with respect to the reference load location. Also, when selecting the reference load location with a positive value of Width specified, then the following two views define the reference load location.
Notice that in the left view, the reference point is on the positive Z wheel track side; whereas in the right view, the reference point is on the least positive X wheel track side.

Specifying Standard AASHTO Loadings

Truck loads specified in AASHTO specifications are also built in to STAAD.

Example 1

```
DEFINE MOVING LOAD
TYPE 1 LOAD 10.0 20.0 –
    15.0 10.0
DISTANCE 5.0 7.5 –
    6.5 WIDTH 6.0
TYPE 2 HS20 0.80 22.0
```

Example 2

When data is provided through an external file called MOVLOAD

```
Data in STAAD input file
UNIT …
DEFINE MOVING LOAD FILE MOVLOAD
TYPE 1 AXLTP1
TYPE 2 AXLTP2 1.25
```
TR.31.2 Definitions for Static Force Procedures for Seismic Analysis

STAAD.Pro offers facilities for determining the lateral loads acting on structures due to seismic forces, using the rules available in several national codes and widely accepted publications. The codes and publications allow for so called equivalent static force methods to be used in place of more complex methods like response spectrum and time history analysis.

Once the lateral loads are generated, the program can then analyze the structure for those loads using the applicable rules explained in the code documents.

Table 236: Codes available in STAAD.Pro with Seismic loads

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Algeria</td>
<td>RPA 99</td>
<td>Règles Parasismiques Algériennes</td>
</tr>
<tr>
<td>Canada</td>
<td>NRC 1995</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td></td>
<td>NRC 2005</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td></td>
<td>NRC 2010</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td>China</td>
<td>GB50011-2001</td>
<td>Code for Seismic Design of Buildings</td>
</tr>
<tr>
<td>Country</td>
<td>Code</td>
<td>Title</td>
</tr>
<tr>
<td>-----------</td>
<td>----------</td>
<td>-----------------------------------------------------------------------</td>
</tr>
<tr>
<td>Colombia</td>
<td>Colombian (on page 2571)</td>
<td><em>Reglamento Colombiano de Construcción Sismo Resistente</em> (NSR-98), <em>Normas Colombianas de Diseño y Construcción</em>, 1998, Asociación Colombiana de Ingeniería Sísmica</td>
</tr>
<tr>
<td></td>
<td>Colombian 2010 (on page 2573)</td>
<td>NSR-10 <em>Reglamento Colombiano Sismo Resistente</em></td>
</tr>
<tr>
<td>India</td>
<td>IS:1893 1984 (on page 2576)</td>
<td>Criteria for Earthquake Resistant Design of Structures</td>
</tr>
<tr>
<td>Japan</td>
<td>AIJ 2006 (on page 2605)</td>
<td>Building Codes Enforcement Ordinance 2006</td>
</tr>
<tr>
<td></td>
<td>NTC (on page 2610)</td>
<td>Reglamento de Construcciones del Distrito Federal de México (Mexico Federal District)</td>
</tr>
<tr>
<td>Turkey</td>
<td>Turkish (on page 2613)</td>
<td>&quot;Specification for Structures to be Built in Disaster Areas Part – III – Earthquake Disaster Prevention&quot; Amended on 2.7.1998, Official Gazette No. 23390</td>
</tr>
</tbody>
</table>
### Related Links

- [M. To add a seismic load definition](on page 840)
- [G.16.2 Seismic Load Generator](on page 2343)

#### TR.31.2.1 RPA (Algerian) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per RPA specifications using a static equivalent approach similar to those outlined by RPA. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in [TR.5 Set Command Specification](on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

#### General Format

```
DEFINE RPA (ACCIDENTAL) LOAD
  rpa-spec
  SELFWEIGHT
  JOINT WEIGHT joint-list
  WEIGHT w
  MEMBER WEIGHT mem-list { UNIT v1 v2 v3 | CON
  ELEMENT WEIGHT
  plate-list PRESS p1
  FLOOR WEIGHT YRANGE ...
```

---

**Country** | **Code** | **Title**
---|---|---


**Technical Reference of STAAD Commands**

**TR.31 Definition of Load Systems**

**Note:** See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

```plaintext
rpa-spec = { A f1 Q f2 RX f3 RZ f4 STYPE f5 CT f6 CRDAMP f7 (PX f8) (PZ f9) }
```

- **A f1**  
  Seismic zone coefficient. Use a fractional value such as 0.08, 0.15, 0.2, 0.3, 0.05, etc., instead of an integer value.

- **Q f2**  
  Importance factor

- **RX f3**  
  Coefficient R for lateral load in X direction – table 4.3

- **RZ f4**  
  Coefficient R for lateral load in Z direction – table 4.3

- **STYPE f5**  
  Soil Profile Type

- **CT f6**  
  Coefficient from table 4.6 of RPA 99

- **CRDAMP f7**  
  Critical damping factor

- **PX f8**  
  Optional Period of structure (in sec) in X direction

- **PZ f9**  
  Optional Period of structure (in sec) in Z direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh method. Used for Y if the SET Z UP command is used.

**Generation of RPA Seismic Load**

General format to provide RPA Seismic load in any load case:

```plaintext
LOAD i
RPA LOAD {X | Y | Z} (f10) (ACC f11)
```

Where:

- **LOAD i**  
  the load case number

- **RPA LOAD {X | Y | Z} f10**  
  factor to multiply horizontal seismic load.

- **ACC f11**  
  multiplying factor for Accidental Torsion, to be used to multiply the RPA accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

**Note:** If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the RPA specifications. The value of the accidental torsion is based on the “center of mass” for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHTS and MEMBER WEIGHTS you have specified.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

**Methodology**

The design base shear is computed in accordance with section 4.2.3 of the RPA 99 code. The primary equation, namely 4-1, as shown below, is checked.
\[ V = \left( A \ D \ Q \right) W / R \]

Where:

- \( W \) = total weight on the structure
- \( A \) = zone coefficient
- \( D \) = average dynamic amplification factor
- \( R \) = lateral R factor
- \( Q \) = structural quality factor

Seismic zone coefficient and parameter values are supplied by the user through the `DEFINE RPA LOAD` command.

Program calculates the natural period of building \( T \) utilizing clause 4.2.4 of RPA 99.

Design spectral coefficient (\( D \)) is calculated utilizing \( T \) as,

\[
D = 2.5\eta \text{ when } 0 \leq T \leq T_2
\]
\[
= 2.5\eta(T_2^2/T)^{2/3} \text{ when } T_2 \leq T \leq 0.3 \text{ sec.}
\]
\[
= 2.5\eta(T_2^3/3)^{2/3}(3/T)^{5/3} \text{ when } T > 0.3 \text{ sec.}
\]

Where:

- \( \eta \) = factor of damping adjustment (Eq. 4.3)
- \( T_2 \) = specific period (Table 4.7)

Total lateral seismic load, \( V \) is distributed by the program among different levels.

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE RPA LOAD` command.

**Example**

```
DEFINE RPA LOAD
A 0.15 Q 1.36 STYP 2 RX 3 RZ 4 CT 0.0032 -
CRDAMP 30 PX .027 PZ 0.025
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
RPA LOAD X 1.0
```

**Related Links**
- [G.16.2 Seismic Load Generator](on page 2343)

**TR.31.2.2 Canadian Seismic Code (NRC) - 1995**

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada - 1995 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in [TR.5 Set](#)).
**Command Specification** (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

### General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE NRC LOAD` command.

```
DEFINE NRC LOAD
  nrc-spec
    SELFWEIGHT
    JOINT WEIGHT joint-list
    WEIGHT w
    MEMBER WEIGHT mem-list
      { UNI v1 v2 v3 | CON }
    ELEMENT WEIGHT plate-list
    PRESS p1
    FLOOR WEIGHT
    YRANGE ...
```

**Note:** See [TR.31.20 UBC 1994 or 1985 Load Definition](on page 2617) for complete weight specification and [TR.32.4 Area, One-way, and Floor Load Specifications](on page 2664) for `FLOOR WEIGHT` input description.

Where:

- `nrc-spec = *{ V f1 ZA f2 ZV f3 RX f4 RZ f5 I f6 F f7 (CT f8) (PX f9) (PZ f10) }`
- `V f1` Zonal velocity ratio per Appendix C
- `ZA f2` Factor for acceleration related seismic zone per Appendix C
- `ZV f3` Factor for velocity related seismic zone per Appendix C
- `RX f4` Force modification factor along X-direction that reflects the capability of a structure to dissipate energy through inelastic behavior. Please refer Table 4.1.9.1B
- `RZ f5` Force modification factor along Z-direction that reflects the capability of a structure to dissipate energy through inelastic behavior. Please refer Table 4.1.9.1B
- `I f6` Seismic importance factor per sentence 10 section 4.1.9.1
- `F f7` Foundation factor conforming to Table 4.1.9.1C and sentence 11 section 4.1.9.1
- `CT f8` Factor to be used to calculate the fundamental period of structure. This is an optional parameter.
- `PX f9` Period of structure (in seconds) in the X-direction. This is an optional parameter.
- `PZ f10` Period of structure (in seconds) in the Z-direction. This is an optional parameter. Used for Y if the `SET Z UP` command is used.
- `w` joint weight associated with list
- `v1, v2, v3` Used when specifying a uniformly distributed load with a value of `v1` starting at a distance of `v2` from the start of the member and ending at a distance of `v3` from the start of the member. If `v2` and `v3` are omitted, the load is assumed to cover the entire length of the member.
**v4, v5**  
Used when specifying a concentrated force with a value of v4 applied at a distance of v5 from the start of the member. If v5 is omitted, the load is assumed to act at the center of the member.

**p1**  
weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (see TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for details of the Floor Load input).

The weights have to be input in the order shown.

---

### Generation of NRC Load

The load so defined as above is applied on the structure in the NRC loadcases. These loadcases have to be the first loadcases in the input file. Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications.

The following general format should be used to generate loads in a particular direction.

```
LOAD i
NRC LOAD { X | Y | Z } (f1)
```

Where:

- **LOAD i**  
  load case number

- **NRC LOAD { X | Y | Z } (f1)**  
  factor to be used to multiply the NRC Load (default = 1.0). May be negative.

**Notes**

1. By providing either PX or PZ or both, you may override the period calculated by STAAD using Rayleigh method. If you do not define PX or PZ, the period for Method 2(b) above will be calculated by the program using Rayleigh method and the stipulations of sentence 7(c) of section 4.1.9.1

2. Some of the items in the output for the NRC analysis are explained below.

   \[ T_a = \text{Time period calculated per sentence 7(a) or 7(b) of section 4.1.9.1} \]

   \[ T_c = \text{Time period calculated per sentence 7(c) of section 4.1.9.1} \]

   **CALC / USED PERIOD**

   The CALC PERIOD is the period calculated using the Rayleigh method. For NRC in the x-direction, the USED PERIOD is PX. For the NRC in the z-direction (or Y direction if SET Z UP is used), the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one utilized to find out the value of S.

3. In the analysis for NRC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).
Methodology

The minimum lateral seismic force or base shear \( V \) is automatically calculated by STAAD using the appropriate equation(s); namely sentence 4, section 4.1.9.1 of NRC.

\[ V = 0.6 \cdot \frac{V_e}{R} \]

Where:

\( V_e \), the equivalent lateral seismic force representing elastic response (per sentence 5, section 4.1.9.1) is given by:

\[ V_e = v \cdot S \cdot I \cdot W \]

Where:

\( v = \) Zonal velocity ratio per appendix C
\( S = \) Seismic Response Factor per table 4.9.1.A
\( I = \) Seismic importance factor per sentence 10 section 4.1.9.1
\( F = \) Foundation factor conforming to Table 4.9.1.C and sentence 11 section 4.1.9.1
\( W = \) Total load lumped as weight per sentence 2 section 4.1.9.1
\( R = \) Force modification factor conforming to Table 4.9.1.B that reflects the capability of a structure to dissipate energy through inelastic behavior.

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. The program calculates the fundamental period \( T \) of the structure by
   a. finding out whether the structure being analysed is a moment resisting frame made primarily of steel or of concrete or it is a structure of any other type. Alternatively, the software uses the optional parameter CT if provided. The calculation is done per sentence 7(a) & 7(b) of section 4.1.9.1.
   b. using the Rayleigh method or using the optional parameters PX, PZ - if provided. The stipulations of sentence 7(c) of section 4.1.9.1 are also considered while calculating.
   c. taking the conservative value of \( T \) between those calculated by methods (a) and (b) above.
2. The program finds out the value of Seismic Response Factor \( S \) per table 4.9.1.A utilizing the values of \( T \) as calculated above and the values of \( ZA \) & \( ZV \) input by the user.
3. The program calculates \( V \) per sentence 4 section 4.1.9.1. \( W \) is obtained from the weight data (SELFWEIGHT, JOINT WEIGHT(s), etc.) provided by the user through the DEFINE NRC LOAD command. The weight data must be in the order shown.
4. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per applicable NRC guidelines like sentence 13(a) section 4.1.9.1.

Example

```
DEFINE NRC LOAD
V 0.2 ZA 4 ZV 4 RX 4 RZ 4 I 1.3 F 1.3 CT 0.35 PX 2 PZ 2
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.8
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```
TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada- 2005 Volume 1 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

STAAD.Pro utilizes the following format to generate the lateral seismic loads.

```plaintext
DEFINE NRC 2005 (ACCIDENTAL) LOAD

nrc-spec

SELFWEIGHT
JOINT WEIGHT joint-list
WEIGHT w
MEMBER WEIGHT mem-list { UNI v1 v2 v3 | CON
ELEMENT WEIGHT plate-list PRESS p1
FLOOR WEIGHT
YRANGE ...
```

**Note:** See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

```plaintext
nrc-spec = { SA1 f1 SA2 f2 SA3 f3 SA4 f4 MVX f5 MVZ f6 JX f7 JZ f8 IE f9 RDX f10 ROX f11 RDZ f12 ROZ f13 SCLASS f14 ( FA f15 ) ( FV f16 ) ( CT f17 ) ( PX f18 ) ( PZ f19 ) }
```

where:

- **SA1 f1** Seismic Data, Sa(0.2), as per Table C-2.
- **SA2 f2** Seismic Data, Sa(0.5), as per Table C-2.
- **SA3 f3** Seismic Data, Sa(1.0), as per Table C-2.
- **SA4 f4** Seismic Data, Sa(2), as per Table C-2.
- **MVX f5** The higher mode factor along the X direction. Refer NRC Table 4.1.8.11.
- **MVZ f6** The higher mode factor along the Z direction. Refer NRC Table 4.1.8.11.
- **JX f7** The numerical reduction coefficient for base overturning moment along the X direction. Refer to NRC Table 4.1.8.11.
The numerical reduction coefficient for base overturning moment along the Z direction. Refer to NRC Table 4.1.8.11.

The earthquake importance factor of the structure. This input is based on Importance Category and ULS / SLS. Refer to NRC Table 4.1.8.11.

The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

The over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

The over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

an integer corresponding to site classes A through E (where 1 = A and 6 = F). $F_a$ and $F_v$ are determined based on Site Class as per NRC Table 4.1.8.4.B and Table 4.1.8.4.C.

Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., $f_{14}$ = 6).

Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., $f_{14}$ = 6).

Optional CT value used in calculating the period of the structure based on the empirical method.

Optional period of the structure (in sec) in the X direction, to be used as fundamental period of the structure. If not entered the value is calculated from the code.

Optional period of the structure (in sec) in the Z direction, to be used as fundamental period of the structure. If not entered the value is calculated from the code.

If the ACCIDENTAL option is specified, the program calculates the accidental torsion per the NRC 2005 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands.

The ACC option along with accidental eccentricity factor (default 0.1 as per NRC 2005) needs to be provided in the NRC seismic primary load case (i.e., NRC LOAD X / Z f1 ACC f3). $f_3$ can be negative.

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the ACCIDENTAL option should not be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the NRC seismic primary load case (i.e., NRC LOAD X / Z f1 DEC f2 ACC f3). For equivalent seismic analysis, $f_2$ is 1 and $f_3$ is 0.1 as per NRC 2005 code. $f_1$ is always positive or zero, however $f_2$ can be negative. If $f_2$ is 0.0, only accidental torsion will be considered for this particular load case.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).
Methodology

The minimum lateral seismic force or base shear \( V \) is automatically calculated by STAAD.Pro using the appropriate equation (s).

\[
V = \frac{S(T_a)M_v I_E W}{R_d R_o}
\]

as per section 4.1.8.11(2) of NBC of Canada 2005, Volume 1

Except that \( V \) shall not be less than:

\[
V_{\text{min}} = \frac{S(2.0)M_v I_E W}{R_d R_o}
\]

and for an \( R_d = 1.5 \), \( V \) need not be greater than:

\[
V_{\text{max}} = \frac{2}{3} \frac{S(0.2) I_E W}{R_d R_o}
\]

(i.e., the upper limit of \( V \))

Description of the terms of the equation to calculate \( V \):

- \( T_a \) is the fundamental lateral period in the direction under consideration and is determined as:
  
  a. For moment-resisting frames that resist 100% of the required lateral forces and where the frame is not enclosed by or adjoined by more rigid elements that would tend to prevent the frame from resisting lateral forces, and is calculated by the empirical formulae as described below provided \( h_n \) is in meter
    
    i. \( 0.085(h_n)^{3/4} \) for steel moment frames,
    
    ii. \( 0.075(h_n)^{3/4} \) for concrete moment frames, or
  
  b. The period is also calculated in accordance with the Rayleigh method but could be overridden by user specified time period (PX, PZ).

  If design spectral acceleration, \( S(T_a) \), calculated considering structural time period calculated based on method (b) is greater than 0.8 time the same calculated considering structural time period calculated based on method (a), the former is used for further calculation. Otherwise, the later time period is used.

  c. Other established methods of mechanics using a structural model that complies with the requirements of Sentence 4.1.8.3.(8), except that
    
    i. for moment resisting frames, \( T_a \) shall not be greater than 1.5 times that determined in Clause (a).
    
    ii. for braced frames, \( T_a \) shall not be greater than 2.0 times that determined in Clause (b).
    
    iii. for shear wall structures, \( T_a \) shall not be greater than 2.0 times that determined in Clause (c), and
    
    iv. for the purpose of calculating the deflections, the period without the upper limit specified is referred from Appendix A.

- \( S(T_a) \) is the design spectral acceleration and is determined as follows, using linear interpolation for intermediate values of \( T_a \):

  \[
  S(T_a) = F_a S_a(0.2) \text{ for } T_a \leq 0.2s
  = F_v S_a(0.5) \text{ or } F_a S_a(0.2), \text{ whichever is smaller for } T_a = 0.5s
  = F_v S_a(1.0) \text{ for } T_a = 1.0s
  = F_v S_a(2.0) \text{ for } T_a = 2.0s
  = F_v S_a(2.0)/2 \text{ for } T_a \geq 4.0s
  \]
The above terms $S_a(0.2)$, $S_a(0.5)$, $S_a(1.0)$ and $S_a(2.0)$ are the Seismic Data and are obtained as user input from the Table C-2.

Based on the above values of $S_a(T_a)$, $F_a$ and $F_v$, the acceleration- and velocity based site coefficients are determined from the Tables 4.1.8.4.B and 4.1.8.4.C, using linear interpolation for intermediate values of $S_a(0.2)$ and $S_a(1.0)$. It is to be mentioned that, these are the user inputs based on the site classes from A to E and the desired $S_a(0.2)$ and $S_a(1.0)$ values as required as per the above equations.

- $M_v$ is the factor to account for higher mode effect on base shear and the associated base overturning moment reduction factor is $J$ which are obtained as user input from the Table 4.1.8.11. To get this higher mode factor ($M_v$) and numerical reduction coefficient for base overturning moment ($J$), you must get the ratios of $S_a(0.2)/S_a(2.0)$ as also the “Type of Lateral Resisting System.”

For values of $M_v$ between fundamental lateral periods, $T_a$ of 1.0 and 2.0 s, the product $S(T_a)$. $M_v$ shall be obtained by linear interpolation.

Values of $J$ between fundamental lateral periods, $T_a$ of 0.5 and 2.0 s shall be obtained by linear interpolation.

- $I_E$ is the earthquake importance factor of the structure and is determined from the Table 4.1.8.5. This is a user input depending on Importance Category and ULS / SLS

- $W$ is the weight of the building and shall be calculated internally using the following formula:

$$W = \sum_{i=1}^{n} W_i$$

Where $W_i$ is the portion of $W$ that is located at or assigned to level $i$.

- $R_d$ is the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9.

- $R_o$ is the over-strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9.

These $R_d$ and $R_o$ values are the user inputs depending on the type of SFRS.

As per 4.1.8.11(6), the total lateral seismic force, $V$, shall be distributed such that a portion, $F_t$, shall be concentrated at the top of the building, where,

$$F_t = 0.07 T_a V$$

but $F_t$ is not greater than 0.25$V$ and $F_t = 0$ when $T_a$ is not greater than 0.7s.

The remainder $(V - F_t)$, shall be distributed along the height of the building, including the top level, in accordance with the following formula [as per section 4.1.8.11(6)]:

$$F_x = \frac{(V - F_t) W_x h_x}{\sum_{i=1}^{n} W_i h_i}$$

where

- $F_x$ = the lateral force applied to level $x$
- $F_t$ = the portion of $V$ to be concentrated at the top of the structure
- $W_i, W_x$ = the portion of $W$ that is located at or assigned to level $i$ or $x$, respectively
- $h_i, h_x$ = the height above the base ($i=0$) to level $i$ or $x$, respectively
- $i = $ is any level in the building (e.g., $i = 1$ for the first level above the base)
- $n = $ is the uppermost level in the main portion of the structure
Example

```
DEFINE NRC 2005 ACC LOAD
SA1 .33 SA2 .25 SA3 .16 SA4 .091 MVX 1.2 MVZ 1.5 JX .7 JZ .5 IE 1.3 -
RDX 4.0 ROX 1.5 RDZ 3.0 ROZ 1.3 SCLASS 4
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

Related Links

- [G.16.2 Seismic Load Generator](on page 2343)
- [V. NRC 2005 Static Seismic](on page 3559)

**TR.31.2.4 Canadian Seismic Code (NRC) - 2010**

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada - 2010 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in [TR.5 Set Command Specification](on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

**General Format**

STAAD.Pro utilizes the following format to generate the lateral seismic loads.

```
DEFINE NRC 2010 (ACCIDENTAL) LOAD

nrc-spec

SELFWEIGHT
JOINT WEIGHT joint-list
WEIGHT w
MEMBER WEIGHT mem-list { UNI v1 v2 v3 | CON
ELEMENT WEIGHT
plate-list PRESS p1
FLOOR WEIGHT
YRANGE ...
```

**Note:** See [TR.31.2.20 UBC 1994 or 1985 Load Definition](on page 2617) for complete weight specification and [TR.32.4 Area, One-way, and Floor Load Specifications](on page 2664) for FLOOR WEIGHT input description.

Where:

```
nrc-spec = { SA1 f1 SA2 f2 SA3 f3 SA4 f4 MVX f5 MVZ f6 I f7 RDX f8 ROX f9 RDZ f10 ROZ f11 SCLASS f12 STX f13 STZ f14 MD f15 (CTX f16 ) (CTZ f17 ) (PX f18 ) (PZ f19 ) (FA f20 ) (FV f21 )
```
where:

**SA1 f1**  Seismic Data, Sa(0.2), as per Table C-2.

**SA2 f2**  Seismic Data, Sa(0.5), as per Table C-2.

**SA3 f3**  Seismic Data, Sa(1.0), as per Table C-2.

**SA4 f4**  Seismic Data, Sa(2), as per Table C-2.

**MVX f5**  The higher mode factor along the X and Z direction. Refer to NRC Table 4.1.8.11.

**MVZ f6**  The higher mode factor along the Z direction. Refer to NRC Table 4.1.8.11.

**I f7**  Earthquake Importance Factor, \( I_e \), of the structure as determined from table 4.1.8.5 in the code. This dependant on the Importance Category and ULS/SLS.

**RDX f8**  The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

**RDZ f10**  The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

**ROX f9**  the over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

**ROZ f11**  the over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

**SCLASS f12**  an integer corresponding to site classes A through E (where 1 = A and 6 = F). \( F_a \) and \( F_v \) are determined based on Site Class as per Table 4.1.8.4.B and Table 4.1.8.4.C.

**STX f13**  Type of lateral resisting system along the X direction. The parameters can take values from 1 to 5:

1 = Moment Resisting Frames
2 = Coupled Walls
3 = Braced Frames
4 = Walls, wall-frame systems
5 = Other systems

**STZ f14**  Type of lateral resisting system along the Z direction. The parameters can take values from 1 to 5:

1 = Moment Resisting Frames
2 = Coupled Walls
3 = Braced Frames
4 = Walls, wall-frame systems
5 = Other systems

**MD f15**  Command to check if the time period calculated is for the purpose of Member Strength or Deflection as per Clause 4.1.8.11.(3).(v)

1 = Member strength
2 = Deflection

**CTX** _f_16  Optional CT value along the X direction to calculate time period.

**CTZ** _f_17  Optional CT value along the Z direction to calculate time period.

**PX** _f_18  Optional Periods of structure (in sec) in the X direction to be used as fundamental period of the structure. If not entered the value is calculated from the code.

**PZ** _f_19  Optional Periods of structure (in sec) in the Z direction to be used as fundamental period of the structure. If not entered the value is calculated from the code.

**FA** _f_20  Optional Short-Period site coefficient at 0.2s.

**Note:** Value must be provided if SCLASS set to F (i.e., _f_12 = 6).

**FV** _f_21  Optional Long-Period site coefficient at 1.0s.

**Note:** Value must be provided if SCLASS set to F (i.e., _f_12 = 6).

If the ACCIDENTAL option is specified, the program calculates the accidental torsion per the NRC 2010 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands.

The ACCIDENTAL option along with accidental eccentricity factor (default 0.1 as per NRC 2010) needs to be provided in the NRC seismic primary load case (i.e., NRC LOAD X / Z _f_1 ACC _f_3). _f_3 can be negative.

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the ACCIDENTAL option should not be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the NRC seismic primary load case (i.e., NRC LOAD X / Z _f_1 DEC _f_2 ACC _f_3). For equivalent seismic analysis, _f_2 is 1 and _f_3 is 0.1 as per NRC 2005 code. _f_1 is always positive or zero, however _f_2 can be negative. If _f_2 is 0.0, only accidental torsion will be considered for this particular load case.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

**Methodology**

The equivalent static force procedure to obtain the base shear is implemented according to section 4.1.8.11, Division B of NBCC 2010.

**Seismic Base Shear**

The minimum lateral earthquake force is calculated according to the following equation (see 4.1.8.11.2)

\[
V = \frac{S(T_a)M_v I_E W}{R_A R_0}
\]

- For walls, coupled walls and wall-frame systems, _V_ shall not be less than \(S(2.0)M_v I_E W/R_A R_0\)
  
- For moment-resisting frames, braced-frames and other systems, _V_ shall not be less than \(S(2.0)M_v I_E W/R_A R_0\)
$M_v$ must be calculated with $T \geq 2.0$

- And for $V > 1.5$, $V$ need not be greater than

$$\frac{2}{3}S(0.2)I_E W \bigg| \left( R_d R_o \right)$$

**Fundamental Period, $T_a$**

The fundamental period, $T_a$, is based on one of the following choices:

- Clause (a): Use 4.1.18.11.3a:
  - i. $0.085(h_n)^{3/4}$ for steel moment frames
  - ii. $0.075(h_n)^{3/4}$ for concrete moment frames
- Clause (b):
  - $0.025h_n$ for braced frames
- Clause (c):
  - $0.05(h_n)^{3/4}$ for shear walls and other structures

where

$$h_n = \text{the height of the building in meters}$$

In the preceding equations, clauses (a), (b) and (c) are implemented except $T_a = 0.1N$. The period is also calculated in accordance with the Rayleigh method.

You may also specify the time period using parameters $PX$ an $PY$. In this case the program checks the following limits:

- Clause d-i: For moment resisting frames: $T_a \leq 1.5 \times$ that determined in clause (a)
- Clause d-ii: For braced frames: $T_a \leq 2.0 \times$ that determined in clause (b)
- Clause d-iii: For shear wall structures: $T_a \leq 2.0 \times$ that determined in clause (c)
- Clause d-iv: For other structures: $T_a \leq$ that determined in clause (c)

However, if the load case is created for drift provisions, (i.e., calculating drift/deflections), the above limits are not checked but instead the following limits are enforced (see Clause d-v):

- $T_a \leq 2.0$ if it is a moment-resisting frame, braced frame, or other system
- $T_a \leq 4.0$ for all others (i.e., walls, coupled walls, and wall frame system)

Also, it is stated in code that these upper limits specified may not be checked for deflection and period calculations.

**Design Spectral Response Acceleration, $S(T)$**

The design spectral acceleration, $S(T)$, is determined as follows, using linear interpolation for intermediate values of $T$:

$$S(T) = \begin{cases} 
  F_v S_a(0.2) & \text{for } T \leq 0.2s \\
  F_v S_a(0.5) & \text{for } T = 0.5s \\
  F_v S_a(1.0) & \text{for } T = 1.0s \\
  F_v S_a(2.0) & \text{for } T = 2.0s \\
  \frac{1}{2} F_v S_a(2.0) & \text{for } T \geq 4.0s 
\end{cases}$$
The above data Sa(0.2), Sa(0.5), Sa(1.0) and Sa(2) are the seismic data and are provided in the parameters SA1, SA2, SA3, and SA4, respectively, from the table C-2.

Based on the above values of Sa(Ta), Fa and Fv, the acceleration- and velocity-based site coefficients are determined from the Tables 4.1.8.4.B and 4.1.8.4.C, using linear interpolation for intermediate values of Sa(0.2) and Sa(1.0). It is to be mentioned that, these are the user inputs based on the site classes from A to E and the desired Sa(0.2) and Sa(1.0) values as required as per the above equations.

**Higher Mode Factor, \( M_v \)**

\( M_v \) is the factor to account for higher mode effect on base shear which is obtained from the Table 4.1.8.11 based on Lateral Resisting System, Sa(0.2), and Sa(2.0). To get this higher mode factor \( (M_v) \), you must evaluate the ratios of \( Sa(0.2)/Sa(2.0) \) as also the “Type of Lateral Resisting System.” You may alternatively directly specify \( M_v \) using the \( MVX \) and \( MVZ \) parameters.

**Force Modification Factors, \( R_d \) and \( R_o \)**

The ductility related force modification factor, \( R_d \), reflects the capability of a structure to dissipate energy through inelastic behavior. The over-strength-related force modification factor, \( R_o \), accounts for the dependable portion of reserve strength in a structure designed according to the provision of Article 4.1.8.9. These values are specified using the \( RDX, ROX, RDZ, \) and \( ROZ \) parameters and depend on the type of SFRS.

**Seismic Weight of the Building, \( W \)**

Calculated by the program as:

\[
W = \sum_{i=1}^{n} W_i
\]

where

\[
W_i = \text{the portion of } W \text{ that is located at or assigned to level } i
\]

**Distribution of Lateral Earthquake Force**

The calculated base shear, \( V \), is distributed over the height of the building by the following equation:

\[
F_x = \frac{(V - F_t)W_xh_x}{\sum_{i=1}^{n} W_ih_i}
\]

where

\[
F_t = \text{Concentrated force applied on the top of the building and it accounts for effects of higher order modes. As per Clause 4.1.8.11(6) } F_t \text{ is equal to } 0.07T_aV, \text{ but } F_t \text{ is not greater than } 0.25V \text{ and } F_t = 0 \text{ when } T_a \text{ is not greater than } 0.7s
\]

\[
W_i = \text{the portion of } W \text{ that is located at or assigned to level } i \text{ or } x
\]

\[
W_x = \text{respectively}
\]

\[
h_i, h_x = \text{height above the base} (i=0) \text{ to level } i \text{ or } x \text{ respectively}
\]

\[
i = \text{any level of the building, } i=1 \text{ for the first floor above the base and } i=n \text{ for the uppermost level in the main portion of the structure}
\]

**Torsional Effect**

Torsional effects are accounted for according to the Clause 4.1.8.11(10):

\[
T_x = F_x(e_x + 0.10D_{nx})
\]

\[
T_x = F_x(e_x - 0.10D_{nx})
\]

where
Natural eccentricity due to center of rigidity and center of mass being at different positions

Plan dimension normal to the direction of ground motion

Note: The overturning moment calculation as given in 4.1.8.11 is not evaluated by the program.
DEFINE MATERIAL

ISOTROPIC MATERIAL1
E 2.5e+007
POISSON 0.17
DENSITY 24

ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2

ISOTROPIC CONCRETE
E 2.17185e+007
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-005
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL

CONSTANTS

MATERIAL MATERIAL1 MEMB 101 TO 136 201 TO 236
MATERIAL STEEL MEMB 301 TO 348

SUPPORTS
1 TO 16 FIXED
CUT OFF MODE SHAPE 10

DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1

JOINT LOAD
17 TO 48 FX 7
49 TO 64 FX 3.5
17 TO 48 FY 7
49 TO 64 FY 3.5
17 TO 48 FZ 7
49 TO 64 FZ 3.5

END DEFINE REFERENCE LOADS

FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3.5
DIA 2 TYPE RIG HEI 7
DIA 3 TYPE RIG HEI 10.5

*** Equivelant Lateral Force Definition ***
DEFINCE NRC 2010 LOAD
SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1.2 MVZ 1.2 -
RDX 1.4 RDZ 3 ROX 1.5 ROZ 1.5 STX 3 STZ 4 MD 1

***********************

*** X-DIRECTION
LOAD 1 FX+TX
NRC LOAD X 1 DEC 1 ACC 0.1
LOAD 2 FX-TX
**Related Links**

- V. NRC 2010 Static Seismic (on page 3569)

**TR.31.2.5 Chinese Static Seismic per GB50011-2001**

This set of commands may be used to define and generate static equivalent seismic loads as per Chinese specifications GB50011-2001. This load uses a static equivalent approach, similar to that found in the UBC. Depending on this definition, equivalent lateral loads will be generated in the horizontal direction(s).

**General Format**

The following general format should be used to generate loads in a particular direction.

```
DEFINE GB (ACCIDENTAL) LOAD
INTENSITY s1 { FREQUENT | RARE } GROUP i1 SCLASS i2 (DAMP f1) (DELN f2) (SF f3)
    (AV f4) (PX f5) (PZ f6)
```

Where:

- **INTENSITY s1** the Fortification Intensity (ref. table 5.1.4-1). Acceptable values are 6, 7, 7A, 8, 8A, or 9.
- **GROUP i1** Design Seismic Group (ref. table 5.1.4-2). Acceptable values are 1, 2, or 3.
- **SCLASS i2** Site-Class (ref. table 5.1.4-2). Acceptable values are 1, 2, 3, or 4.
- **DAMP f1** damping ratio (default = 0.05 for 5% damping)
DELN \( f_2 \) \( \delta_n \), Additional seismic action factor at the top of the building (default as calculated from Table 5.2.1)

SF \( f_3 \) Shear Factor \( \lambda \), Minimum seismic shear factor of the floor (default as calculated from Table 5.2.5)

AV \( f_4 \) maximum value of vertical seismic influence coefficient \( \alpha_{(v,max)} \) (default=0.0) (ref. section 5.3)

Frequency of seismic action, as specified by either FREQUENT or RARE (ref. table 5.1.2-2)

As a fraction of total vertical load is to be considered such as 0.75Geq, specify the product of the factor on maximum horizontal seismic influence factor and factor of total gravity load as \( f_4 \). For instance,

- \( \alpha_{(v,max)} = 0.65\alpha_{\text{max}} \)

And

- \( G_{v,\text{eq}} = 0.75Geq \)
- Specify \( f_4 \) as (0.65*0.75) i.e., equal to 0.4875

PX \( f_5 \) optional time period along the X direction

PZ \( f_6 \) optional time period along the Z direction

**Generation of GB50011 Seismic Load**

To apply the load in any load case, following command would be used

```
LOAD CASE  i
GB LOAD { X | Y | Z } (f7) (ACC f8)
```

Where:

- LOAD \( i \) load case number
- GB LOAD \{ X | Y | Z \} \( f_7 \) optional factor to multiply horizontal seismic load.
- ACC \( f_8 \) multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

**Note:** If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the GB specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

The seismic load generator can be used to generate lateral loads in the X and Z directions for Y up and the X and Y directions for Z up; where Y up or Z up is the vertical axis parallel to the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)).

**Note:** All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.
This method of seismic load generation is limited in use to buildings not taller than 40 meters, with deformations predominantly due to shear, and a rather uniform distribution of mass and stiffness in elevation. Alternately, for buildings modeled as a single-mass system, a simplified method such as this base shear method, may be used.

Gravity Loads for Design

In the computation of seismic action, the representative value of gravity load of the building shall be taken as the sum of characteristic values of the weight of the structure and members plus the combination values of variable loads on the structure. The combination coefficients for different variable loads shall be taken from the following table.

**Table 237: Combinations of different load effects per GB50011-2001**

<table>
<thead>
<tr>
<th>Type of Variable</th>
<th>Land Combination coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>Snow load</td>
<td>0.5</td>
</tr>
<tr>
<td>Dust load on roof</td>
<td>0.5</td>
</tr>
<tr>
<td>Live load on roof</td>
<td>Not considering</td>
</tr>
<tr>
<td>Live load on the floor, calculated according to actual state</td>
<td>1.0</td>
</tr>
<tr>
<td>Live load on the floor, calculated according to equivalent uniform state</td>
<td>Library, archives</td>
</tr>
<tr>
<td>Other civil buildings</td>
<td>0.5</td>
</tr>
<tr>
<td>Gravity for hanging object of crane</td>
<td>Hard hooks</td>
</tr>
<tr>
<td></td>
<td>Soft hooks</td>
</tr>
</tbody>
</table>

Seismic Influence Coefficient

This shall be determined for building structures according to the Intensity, Site-class, Design seismic group, and natural period and damping ratio of the structure. The maximum value of horizontal seismic influence coefficient shall be taken from Table 2.2; the characteristic period shall be taken as Table 2.3 according to Site-class and Design seismic group, that shall be increased 0.05s for rarely earthquake of Intensity 8 and 9.

**Table 238: Earthquake influence per GB50011-2001**

<table>
<thead>
<tr>
<th>Earthquake influence</th>
<th>Intensity 6</th>
<th>Intensity 7</th>
<th>Intensity 8</th>
<th>Intensity 9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequent earthquake</td>
<td>0.04</td>
<td>0.08 (0.12)</td>
<td>0.16 (0.24)</td>
<td>0.32</td>
</tr>
<tr>
<td>Rarely earthquake</td>
<td>-</td>
<td>0.50 (0.72)</td>
<td>0.90 (1.20)</td>
<td>1.40</td>
</tr>
</tbody>
</table>
Note: The values in parenthesis are separately used for where the design basic seismic acceleration is 0.15g and 0.30g.

Table 239: Earthquake group per GB50011-2001

<table>
<thead>
<tr>
<th>Earthquake Group</th>
<th>Site class</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>I</td>
</tr>
<tr>
<td>1</td>
<td>0.25</td>
</tr>
<tr>
<td>2</td>
<td>0.30</td>
</tr>
<tr>
<td>3</td>
<td>0.35</td>
</tr>
</tbody>
</table>

Calculation of Seismic Influence Coefficient

The design base shear is computed in accordance with the equations shown below.

The damping adjusting and forming parameters on the building seismic influence coefficient curve (Fig.2.1) shall comply with the following requirements:

1. The damping ratio of building structures shall select 0.05 except otherwise provided, the damping adjusting coefficient of the seismic influence coefficient curve shall select 1.0, and the coefficient of shape shall conform to the following provisions:
   a. Linear increase section, whose period (T) is less than 0.1 s;
   b. Horizontal section, whose period form 0. is thought to characteristic period, shall select the maximum value (α_max);
   c. Curvilinear decrease section, whose period from characteristic period thought to 5 times of the characteristic period, the power index (γ) shall choose 0.9.
   d. Linear decrease section, whose period from 5 times characteristic period thought to 6s, the adjusting factor of slope (η_1) shall choose 0.02.

![Figure 304: Seismic influence coefficient curve](image)

2. When the damping adjusting and forming parameters on the seismic influence coefficient curve shall comply with the following requirements:
a. The power index of the curvilinear decreased section shall be determined according to the following equation E2.1

\[ \gamma = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} \]  

where
\[ \gamma \] = the power index of the curvilinear decrease section
\[ \zeta \] = the damping ratio

b. The adjusting factor of slope for the linear decrease section shall be determined from following equation:

\[ \eta_1 = 0.02 + (0.05 - \zeta)/8 \]  

where
\[ \eta_1 \] = the adjusting factor of slope for the linear decrease section, when it is less than 0, shall equal 0.

c. The damping adjustment factor shall be determined according to the following equation:

\[ \eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} \]  

where
\[ \eta_2 \] = the damping adjustment factor, when it is smaller than 0.55 shall equal 0.55.

Calculation of Horizontal Seismic Action

When the base shear force method is used, only one degree of freedom may be considered for each story; the characteristic value of horizontal seismic action of the structure shall be determined by the following equations (Fig. 2.2):

\[ F_{Ek} = \alpha_1 G_{eq} \]  

E2.4

\[ F_i = \sum_{j=1}^{n} \frac{G_j H_j}{G_i H_i} F_{Ek}(1 - \delta_n) \]  

E2.5

\[ \Delta F_n = \delta_n F_{Ek} \]  

E2.6

Figure 305: Calculation of horizontal seismic action
where

\[ F_{Ek} = \] characteristic value of the total horizontal seismic action of the structure

\[ \alpha_1 = \] horizontal seismic influence coefficient corresponding to the fundamental period of the structure, which shall be determined by using Clause 2.3. For multistory masonry buildings and multi-story brick buildings with bottom-frames or inner-frames, the maximum value of horizontal seismic influence coefficient should be taken.

\[ G_{eq} = \] equivalent total gravity load of a structure. When the structure is modeled as a single-mass system, the representative value of the total gravity load shall be used; and when the structure is modeled as a multi-mass system, the 85% of the representative value of the total gravity load may be used.

\[ F_i = \] characteristic value of horizontal seismic action applied on mass \( i \)th level.

\[ G_i, G_j = \] representative values of gravity load concentrated at the masses of \( i \)th and \( j \)th respectively, which shall be determined by Clause 2.1.

\[ H_i, H_j = \] calculated height of \( i \)th and \( j \)th from the base of the building respectively.

\[ \delta_n = \] additional seismic action factors at the top of the building; for multi-story reinforced concrete buildings, it may be taken using Table 2.4; for multi-story brick buildings with inner-frames, a value of 0.2 may be used; no need to consider for other buildings

\[ \Delta F_n = \] additional horizontal seismic action applied at top of the building.

Table 2.4 Additional seismic action factors at top of the building

<table>
<thead>
<tr>
<th>Structures</th>
<th>Intensity 7</th>
<th>Intensity 8</th>
<th>Intensity 9</th>
</tr>
</thead>
<tbody>
<tr>
<td>structures with obvious torsion effect or fundamental period is less than 3.5s</td>
<td>0.16 (0.024)</td>
<td>0.032 (0.048)</td>
<td>0.064</td>
</tr>
<tr>
<td>Structures with fundamental period greater than 5.0s</td>
<td>0.012 (0.018)</td>
<td>0.024 (0.032)</td>
<td>0.040</td>
</tr>
</tbody>
</table>

Note: \( T \), is the fundamental period of the structure.

The horizontal seismic shear force at each floor level of the structure shall comply with the requirement of the following equation:

\[ V_{Eki} > \lambda \sum_{j=1}^{n} G_j E2.7 \]

where

\[ V_{Eki} = \] the floor \( i \)th shear corresponding to horizontal seismic action characteristic value.

\[ \lambda = \] Shear factor, it shall not be less than values in Table 2.5; for the weak location of vertical irregular structure, these values shall be multiplied by the amplifying factor of 1.15.

\[ G_j = \] the representative value of gravity load in floor \( j \)th of the structure.
Note:
1. The values may be selected through interpolation method for structures whose fundamental period is between 3.5s and 5s.
2. Values in the brackets are used at the regions with basic seismic acceleration as 0.15g and 0.30g respectively.

Calculation of vertical seismic action
For tall buildings for Intensity 9, the characteristic value of vertical seismic action shall be determined by the following equations (figure 2.3). The effects of vertical seismic action at floor level may be distributed in proportion of representative value of gravity load acting on the members, which should multiply with the amplified factor 1.5:

\[
F_{Evk} = \alpha_{v\text{max}} G_{eq}
\]

\[
F_i = \frac{G_i H_i}{\sum_{j=1}^{n} G_j H_j} F_{Evk}
\]

where

- \( F_{Evk} \) = characteristic value of the total vertical seismic actions of the structure.
- \( F_v \) = characteristic value of vertical seismic action at the level of mass \( i \)th.
- \( \alpha_{v\text{max}} \) = maximum value of vertical seismic influence coefficient, which may be taken as 65% of the maximum value of the horizontal seismic influence coefficient.
- \( G_{eq} \) = equivalent total gravity load of the structure, which may be taken as 75% of the representative value of the total gravity load of the structure.

\[\text{Figure 306: Sketch for the computation of vertical seismic action}\]

Complementarities
1. Structures having the oblique direction lateral-force-resisting members and the oblique angel to major orthogonal axes is greater than 150, the horizontal seismic action along the direction of each lateral-force-resisting member shall be considered respectively. So we could consider this though the item, the action of the oblique member could be multiplied by this factor as design force.
2. Eccentricity: similar to UBC code. The eccentricity value of gravity center on each floor should be \( e_i = \pm 0.05 L_i \).
where

\[ e_i = \text{Eccentricity value of gravity center on } i^{\text{th}} \text{ floor.} \]
\[ L_i = \text{maximum width of calculated story of the building.} \]

3. Structures having obviously asymmetric mass and stiffness distribution, the torsion effects caused by both two orthogonal horizontal direction seismic action shall be considered; and other structures, it is permitted that a simplified method, such as adjusting the seismic effects method, to consider their seismic torsion effects.

Example Output

*****************************************************************************
* *
* EQUIV. SEISMIC LOADS AS PER SEISMIC DESIGN CODE FOR BUILDINGS *
* (GB50011-2001) OF CHINA ALONG X *
* T CALCULATED = 0.252 SEC. T USER PROVIDED = 1.200 SEC. *
* T USED = 1.200 SEC. *
* MAX. HORIZONTAL SEISMIC INFLUENCE COEFFICIENT = 0.240 *
* CHARACTERISTIC PERIOD = 0.750 SEC. *
* DAMPING RATIO = 0.030 POWER INDEX (GAMMA) = 0.931 *
* DAMPING ADJUSTMENT FACTOR (ETA2) = 1.180 *
* ADJUSTING FACTOR (ETA1) = 0.022 *
* HORIZONTAL SEISMIC INFLUENCE COEFFICIENT (ALPHA1) = 0.183 (18.288%) *
* MINIMUM SHEAR FACTOR AS PER SEC. 5.2.5 (LAMBDA) = 0.050 ( 5.000%) *
* TOTAL HORIZONTAL SEISMIC ACTION = *
* = 0.183 X 285.529 = 52.218 KIP *
* DESIGN BASE SHEAR = 0.750 X 52.218 *
* = 39.164 KIP *
* ADDITIONAL SEISMIC ACTION FACTOR (DELTAN) = 0.020 *
* VERTICAL SEISMIC INFLUENCE COEFFICIENT (ALPHA,VMAX) = -0.108 *
* TOTAL VERTICAL SEISMIC ACTION = *
* = -0.108 X 285.529 = -30.837 KIP *
* TOTAL DESIGN VERTICAL LOAD = 0.750 X -30.837 *
* = -23.128 KIP *
* *
*****************************************************************************

CHECK FOR MINIMUM LATERAL FORCE AT EACH FLOOR [GB50011-2001:5.2.5]
LOAD - 1 FACTOR - 0.750
FLOOR LATERAL GRAVITY LAMBDA LAMBDA ADJUSTMENT
HEIGHT (KIP ) LOAD (KIP ) LOAD (KIP ) (%) MIN (%) FACTOR
------------------------------------------ --------- ---------  -----------
30.000 23.406 79.045 29.61 5.00 1.00
20.000 19.208 180.888 10.62 5.00 1.00
10.000 9.604 282.731 3.40 5.00 1.47
JOINT LATERAL TORSIONAL VERTICAL LOAD - 1
LOAD (KIP ) MOMENT (KIP -FEET) LOAD (KIP ) FACTOR - 0.750

*****************************************************************************

17 FX 0.541 MY 0.000 FY -0.221
18 FX 0.663 MY 0.000 FY -0.271
19 FX 0.663 MY 0.000 FY -0.271
20 FX 0.541 MY 0.000 FY -0.221
21 FX 0.663 MY 0.000 FY -0.271
22 FX 0.785 MY 0.000 FY -0.321
23 FX 0.785 MY 0.000 FY -0.321
24 FX 0.663 MY 0.000 FY -0.271
25 FX 0.663 MY 0.000 FY -0.271
26 FX 0.785 MY 0.000 FY -0.321
27 FX 0.785 MY 0.000 FY -0.321

STAAD.Pro 2570 User Manual
Related Links

- M. To add a seismic load definition (on page 840)
- G.16.2 Seismic Load Generator (on page 2343)

**TR.31.2.6 Colombian NSR-98 Seismic Load**


This implementation uses a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). The load is based on

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

**General Format**

```
DEFINE COLOMBIAN (ACCIDENTAL) LOAD

ZONE f1 ( I f2 S f3)

SELFWEIGHT
JOINT WEIGHT joint-list
WEIGHT w
MEMBER WEIGHT mem-list { UNI v1 v2 v3 | CON
ELEMENT WEIGHT plate-list PRESS p1
FLOOR WEIGHT
YRANGE ...
```

**Note:** See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

- **ZONE f1** Seismic Risk factor
- **I f2** Soil Site Coefficient
- **S f3** Coefficient of Importance
Generation of NSR Seismic Load

General format to provide Colombian Seismic load in any load case:

```
LOAD i

COLOMBIAN LOAD {X | Y | Z} (f4) (ACCIDENTAL f5)
```

Where:

- **LOAD i**: load case number
- **COLOMBIAN LOAD {X | Y | Z}**: factor to multiply horizontal seismic load
- **f4**: multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands specified for this load definition.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

### Example

Using NSR-98:

```
DEFINE COLOMBIAN LOAD
ZONE 0.38 1.0 S 1.5
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
COLOMBIAN LOAD X
```

### Methodology for NSR-98

Seismic zone coefficient and parameter values are supplied by the user through the `DEFINE COLOMBIAN LOAD` command.

The program calculates the natural period of building T utilizing clause 1628.2.2 of UBC 1994.

Design spectral coefficient, $S_a$, is calculated utilizing $T$ as:

$$
S_a = \begin{cases} 
A_a I(1.0 + 5.0T) \text{ when } 0 \leq T \leq 0.3 \text{ sec} \\
2.5A_a I \text{ when } 0.3 < T \leq 0.48 \text{ sec} \\
1.2A_a S_f \frac{T}{T^3} \text{ when } 0.48 < T < 2.4 \text{ sec} \\
A_a I / 2 \text{ when } 2.4 \text{ sec} < T
\end{cases}
$$

where

- $A_a = \text{Seismic Risk factor}$
Soil Site Coefficient

Coefficient of Importance

Base Shear, \( V_s \) is calculated as:

\[
V_s = W \times S_a
\]

where

\( W \) = Total weight on the structure

The total lateral seismic load, \( V_s \), is then distributed by the program among different levels as:

\[
F_x = C_{vx} \times V_s
\]

where

\[
C_{ux} = \frac{W_x h_x K}{\sum_{i=1}^{n} (W_i h_i K)}
\]

\( W_x \) = Weight at the particular level

\( h_x \) = Height of that particular level

\( K \) = 1.0 when \( T \leq 0.5 \text{ sec} \)

\[
0.75 + 0.5T \text{ when } 0.5 < T \leq 2.5 \text{ sec}
\]

\[
2.0 \text{ when } T > 2.5 \text{ sec}
\]

Related Links
- G.16.2 Seismic Load Generator (on page 2343)

**TR.31.2.7 Colombian NSR-10 Seismic Load**

Used to define and generate static equivalent seismic loads as the 1998 edition of the Colombian seismic code, NSR-10 Reglamento Colombiano Sismo Resistente.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

**General Format**

```plaintext
DEFINE COLOMBIAN 2010 (ACCIDENTAL) LOAD

AA f_1 AV f_2 FA f_3 FV f_4 I f_5 ( CT f_6 ) ( PX f_7 ) ( PZ f_8 ) ( ALPHA f_9 )

SELFWEIGHT
JOINT WEIGHT joint-list
MEMBER WEIGHT weight
MEMBER list
UNI v_1 v_2 v_3 | CON
ELEMENT WEIGHT
plate-list PRESS p_1
FLOOR WEIGHT
YRANGE ...
```
Note: See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

**AA** \( f_1 \) coefficient representing effective peak horizontal acceleration for design (Table A.2.4-3 NSR-10). Must be within the range of 0 to 0.5 (inclusive) or the output will indicate an error.

**AV** \( f_2 \) coefficient representing the effective peak horizontal velocity for design (Table A.2.4-4 NSR-10). Must be within the range of 0 to 0.5 (inclusive) or the output will indicate an error.

**FA** \( f_3 \) dimensionless amplification factor that affects the acceleration in the short periods due to site effects (Table A.2.4-3 NSR-10). Must be within the range of 0 to 3.5 (inclusive) or the output will indicate an error.

**FV** \( f_4 \) dimensionless amplification factor that affects the acceleration in the intermediate periods due to site effects (Table A.2.4-4 NSR-10). Must be within the range of 0 to 3.5 (inclusive) or the output will indicate an error.

**I** \( f_5 \) Importance factor (Table A.2.5-1 NSR-10). Must be within the range of 1 to 1.5 (inclusive) or the output will indicate an error.

**CT** \( f_6 \) Optional CT value to calculate time period. (Table A.4.2-1 NSR-10)

**PX** \( f_7 \) Optional Period of structure (in sec) in X-direction to be in lieu of fundamental period of the structure

**PZ** \( f_8 \) Optional Period of structure (in sec) in Z-direction to be in lieu of fundamental period of the structure

**ALPHA** \( f_9 \) Exponent used in calculating the approximate period, \( T_a \) (Table A.4.2-1 NSR-10).

### Generation of NSR Seismic Load

General format to provide Colombian Seismic load in any load case:

\[
\text{LOAD \ } i \text{ \ COLOMBIAN \ LOAD } \{X \mid Y \mid Z\} \ (f_{10}) \ (\text{ACCIDENTAL} \ f_{11})
\]

Where:

**LOAD \ j** load case number

**COLOMBIAN LOAD \{X \mid Y \mid Z\} \ f_{10}** factor to multiply horizontal seismic load

**ACCIDENTAL \ f_{11}** multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands specified for this load definition.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).
Example

Using NSR-10:

```
DEFINE COLOMBIAN 2010 LOAD
AA 0.2 AV 0.15 FA 0.85 FV 1.01 I 1.12
JOINT WEIGHT
6 TO 15 21 TO 30 36 TO 45 WEIGHT 10 LOAD 1
COLOMBIAN LOAD X 1.2
```

Methodology for NSR-10

The Rayleigh calculated time period, $T$, is used to evaluate the design spectral coefficient, $S_a$, as:

$$ S_a = \begin{cases} 
2.5 A_a F_a I (0.4 + 0.6 \frac{T}{T_0}) & \text{when } T < T_0 \\
2.5 A_a F_a I & \text{when } T_0 \leq T < T_C \\
1.2 A_v F_v T_L \frac{I}{T^2} & \text{when } T_C \leq T < T_L \\
1.2 A_v F_v T_L \frac{I}{T^2} & \text{when } T > T_L 
\end{cases} $$

where

- $T_0 = 0.1 \frac{A_v F_v}{A_a F_a}$
- $T_C = 0.48 \frac{A_v F_v}{A_a F_a}$
- $T_L = 2.4 F_v$
- $A_a$ = Seismic Risk factor
- $S$ = Soil Site Coefficient
- $I$ = Coefficient of Importance

Base Shear, $V_s$ is calculated as

$$ V_s = S_a W $$

where

- $W$ = Total weight on the structure

The horizontal seismic force, $F_x$, at any level $x$ is calculated by the program as:

$$ F_x = V_x C_{vx} $$

where

- $C_{vx} = \frac{W_x F_x k}{\sum[W_x F_x k]}$
- $W_x$ = Weight at the particular level
- $k$ = the exponent related to the fundamental period, $T$, of the building:
  - $k = 1.0$ when $T \leq 0.5$ sec
  - $k = 0.75 + 0.5T$ when $0.5 \leq T < 2.5$ sec
  - $k = 2.0$ when $2.5$ sec $\geq T$
TR.31.2.8 IS:1893 - 1984 Code - Lateral Seismic Load

This feature enables you to generate seismic loads using a static equivalent approach per the IS:1893 - 1984 specifications, *Earthquake Resistant Design of Structures*.

**General Format**

```
DEFINE 1893 ( ACCIDENTAL ) LOAD
ZONE f1 K f2 I f3 B f4 PX f5 PZ f6
SELFWEIGHT
JOINT WEIGHT
joint-list WEIGHT w
MEMBER WEIGHT
mem-list { UNI v1 v2 v3 | CON v4 v5 }
REFERENCE LOAD { X | Y | Z }
Ri1 f18
```

where

**Note:** If both mass table data (SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT options) and a REFERENCE LOAD are specified, these will be added algebraically for a combined mass.

where:

- **ZONE f1**: Seismic zone coefficient.
- **K f2**: Performance factor.
- **I f3**: Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance.
- **B f4**: Soil interaction factor.
- **PX f5**: Optional period in the X direction.
- **PZ f6**: Optional period in the Z direction.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).


This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach. Both Part 1 (2002) for building structures and Part 4 (2005) for industrial and stack-like structures are available.
The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads. This facility has not been developed for cases where the Z axis is set to be the vertical direction (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)).

**General Format**

```plaintext
DEFINE 1893 ( ACCIDENTAL ) LOAD ( PART4 )

ZONE f1 { 1893-spec-part1 | 1893-spec-part4 }

SELFWEIGHT

JOINT WEIGHT

joint-list WEIGHT w

MEMBER WEIGHT

mem-list { UNI v1 v2 v3 | CON v4 v5 }

REFERENCE LOAD { X | Y | Z }

Ri1 f18

(CHECK SOFT STORY)
```

where

1893-spec-part1 = RF f2 I f3 { SS f4 | SA f11 } (ST f5) ( { DM f6 | DF f12 } ) (PX f7) (PZ f8) ( { DT f9 | GL f10 } )

1893-spec-part4 = RF f2 I f3 { SS f4 | SA f11 } ST f5 ( { DM f6 | DF f12 } ) (PX f7) (PZ f8) ( { DT f9 | GL f10 } ) (CS f13) (AX f14) (ES f15) CV f16 DV f17

**Note:** If both mass table data (SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT options) and a REFERENCE LOAD are specified, these will be added algebraically for a combined mass.

where

**ZONE f1**  Seismic zone coefficient. Refer to Table 2 of IS:1893 (Part 1)-2002.

**RF f2**  Response reduction factor. Refer Table 7 of IS: 1893 (Part 1) -2002 or Table 3 of IS: 1893 (Part 4) -2005.

**I f3**  Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance. Refer Table 6 of IS: 1893(Part 1)-2002 or Table 2 of IS: 1893 (Part 4)-2005.

**SS f4**  Rock or soil sites factor. Depending on type of soil, average response acceleration coefficient $S_a/g$ is calculated corresponding to 5% damping. Refer Clause 6.4.5 of IS: 1893 (Part 1) -2002 or Clause 8.3.2 of IS: 1893 (Part 4) -2005.

- 1 = hard soil
- 2 = medium soil
- 3 = soft soil

**Note:** Use either SS or SA to specify site conditions. If both parameters are specified, SS is ignored.
For IS 1893 Part 1, the program will calculate natural period as per Clause 7.6 of IS:1893(Part 1)-2002.

1 = RC frame building
2 = Steel frame building
3 = All other buildings

For IS1893 Part 4, the program will calculate period as per Clause 14.1 of IS:1893(Part 4)-2005 for stack-like structures. For Category 1 Industrial Structures base shear is calculated as twice the base shear of other structures as per Clause 8.3 of IS:1893(Part 4)-2005.

1 = Category 1 industrial structure (Part 4)
3 = All other industrial structures
5 = stack-like structures

Note: This parameter is optional for Part 1, but is required for Part 4.

Damping ratio to obtain multiplying factor for calculating $S_a/g$ for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0. Refer Table 3 of IS:1893(Part 1)-2002.

Note: Use either DM or DF to specify damping. If both parameters are specified, DM is ignored.

This should be a value between 0 (zero) and 1.0, inclusive. For example, 7% damping should be specified as 0.07.

Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate $S_a/g$ for generation of seismic load along X direction.

Note: See Note 'b' below.

Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate $S_a/g$ for generation of seismic load along Z direction.

Note: See Note 'b' below.

Depth of foundation below ground level. It should be defined in current units. If the depth of foundation is 30 m or more, the value of $A_h$ is taken as half the value obtained. If the foundation is placed between the ground level and 30 m depth, this value is linearly interpolated between $A_h$ and $0.5A_h$.

Note: Use either DT or GL to specify foundation depth. If both parameters are specified, DT is ignored.

Y coordinate of ground level. A reduced lateral force is applied to levels below this height, per Clause 6.4.4.

Used for Y if the SET Z UP command is used.

Average response spectral acceleration coefficient corresponding to site specific spectra.

Multiplying factor for calculating $S_a/g$. 


CS \textit{f13} Coefficient as given in Table 6 of IS:1893(Part 4)-2005. Valid only for stack-like structures for calculating fundamental time period per Cl. 14.1.


CV \textit{f16} Coefficient of shear force given in Table 6 of IS:1893(Part 4)-2005. Required for stack-like structures (ST 5).

DV \textit{f17} Distribution factor for shear force given in Table 11 of IS:1893(Part 4)-2005. Required for stack-like structures (ST 5).

UNI \textit{v1, v2, v3} Used when specifying a uniformly distributed load with a value of \textit{v1} starting at a distance of \textit{v2} from the start of the member and ending at a distance of \textit{v3} from the start of the member. If \textit{v2} and \textit{v3} are omitted, the load is assumed to cover the entire length of the member.

CON \textit{v4, v5} Used when specifying a concentrated force with a value of \textit{v4} applied at a distance of \textit{v5} from the start of the member. If \textit{v5} is omitted, the load is assumed to act at the center of the member.

\textit{R11 f18} Identification number of a previously defined reference load case. See TR.31.6 Defining Reference Load Types (on page 2642)

This is followed by the magnification factor, \textit{f18} (required for reference loads). The default value is 1.0.

\begin{center}
\textbf{Note:} For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).
\end{center}

\textbf{Notes}

\textbf{a.} If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the IS 1893 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands you have specified. The ACC option along with accidental eccentricity factor (generally 0.05 as per IS 1893 code) needs to be provided in the 1893 seismic primary load case (i.e., \texttt{1893 LOAD X / Z f1 DEC f2 ACC f3}). \textit{f2} can be negative. See TR.32.12.2.2 Generation of IS:1893 Seismic Load (on page 2778)

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the ACCIDENTAL option should not be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the 1893 seismic primary load case (i.e., \texttt{1893 LOAD X / Z DEC f2 ACC f3 DEC f2 ACC f3}). For equivalent seismic analysis, \textit{f2} is 1.5 and \textit{f3} is 0.05 as per IS 1893 code. \textit{f1} is always positive or zero, however \textit{f2} can be negative. If \textit{f2} is 0.0, only accidental torsion will be considered for this particular load case.

\textbf{b.} By default STAAD calculates natural periods of the structure in both X and Z directions respectively which are used in calculation for base shear. If PX and PZ are included, the program will consider these values for calculation of average response acceleration coefficient. If ST is used instead of PX and PZ values, then the program will calculate natural period depending upon the empirical expression given in IS: 1893 (Part 1)-2002 or IS: 1893 (Part 4)-2005.

\textbf{c.} In the case where no rigid floor diaphragm is present, STAAD identifies columns and shear walls (without openings) as vertical components for the purpose of computing lateral stiffness of the story.

The lateral stiffness of a column is calculated as:
The lateral stiffness for a shear wall (without opening) is calculated as:

\[
\frac{1}{12EI + \frac{1.2Ph}{12EI}}
\]

Which is the summation of inverse of flexural stiffness and inverse of shear stiffness, obtained as deflection of a cantilever wall under a single lateral load, P, at its top.

where

- \(h\) = height
- \(A\) = cross-sectional area
- \(G\) = shear modulus of the wall

The summation of lateral stiffnesses of all columns and shear walls at a particular floor level constitutes the total lateral stiffness of that particular story or floor level. The program checks for a soft story of a building along both global X and Z directions respectively. This computation is valid only for those structures whose floors are treated as rigid diaphragm.

**Methodology**

The design base shear is computed by STAAD.Pro for building structures as per IS: 1893 (Part 1) 2002 equation 7.5.3 or for industrial structures as per (Part 4) 2005:

\[
V = A_h W
\]

Where:

\[
A_h = \frac{Z \cdot I \cdot S_a}{2Rg}
\]

When site specific spectra is used per IS 1893 (Part 4) 2005, then:

\[
A_h = \frac{I \cdot S_a}{Rg}
\]

For stack-like structures, the design base shear is computed as per IS 1893 (Part 4) 2005 as:

\[
V = C_vA_h W \cdot D_v
\]

**Note:** All symbols and notations in the above equation are as per IS: 1893(Part 1) 2002 and IS: 1893 (Part 4) 2005.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads:

1. You provide seismic zone coefficient and desired 1893 specs through the `DEFINE 1893 LOAD` command. Use the `PART 4` command option to specify using IS: 1893 (Part 4) 2005.
2. The program calculates the structure period, T.
3. The program calculates \(S_a/g\) utilizing T.
4. The program calculates \(V\) from the above equation. \(W\) is obtained from mass table data entered via `SELFWEIGHT`, `JOINT WEIGHT(s)`, `MEMBER WEIGHT(S)`, and/or `REFERENCE LOAD` you provide through the `DEFINE 1893 LOAD` command.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893 procedures.

See TR.32.12 Generation of Loads (on page 2771) for additional information.

**Soft Story Checking**

**Tip:** When a rigid floor diaphragm (on page 2526) is used, a soft story check may be initiated within that command without some of the limitations imposed on soft story checking in a seismic load.

As per the IS1893-2002 code Clause 7.1, to perform well during an earthquake a building must have simple and regular configuration, adequate lateral strength, stiffness and ductility. This is because a building with simple regular geometry and uniformly distributed mass and stiffness in plan as well as in elevation, will suffer much less damage than buildings with irregular configurations.

According to this standard, a building can be considered irregular, if at least one of the conditions given in Table 4 - Plan Irregularities and Table 5 - Vertical Irregularities, of IS1893-2002 is applicable.

For IS 1893 2002, STAAD.Pro has implemented the methodology to find vertical stiffness irregularities, as given in IS 1893-2002 Table 5 Sl No. (1) i) a) and Sl No. (1) i) b), in the form of soft story checking.

- **Stiffness Irregularities:** Soft Story – As per this provision of the code, a soft story is one in which the lateral stiffness is less than 70 percent of that in the story above or less than 80 percent of the average lateral stiffness of the three story above.

- **Stiffness Irregularities:** Extreme Soft Story – As per this provision of the code, a extreme soft story is one in which the lateral stiffness is less than 60 percent of that in the story above or less than 70 percent of the average lateral stiffness of the three story above.

Thus, if any story of a building is found to be soft or extremely soft, the building is likely to suffer much damage in an earthquake than a similar type of building but has more regular vertical stiffness.

**Note:** STAAD.Pro identifies column and shear wall (without opening) as vertical component for the purpose of computing lateral stiffness of the story. The vertical stiffness of a column is calculated as \( \frac{12EI}{L^3} \) where E is the Young’s modulus, I is the moment of inertia and L is the length of the column respectively and that for a shear wall (without opening) is calculated as \( \frac{Ph^3}{3EI} + \frac{1.2Ph}{AG} \) (i.e., summation of flexural stiffness and shear stiffness, obtained as deflection of a cantilever wall under a single lateral load P at its top) where h is the height, A is the cross-sectional area, and G is the shear modulus of the wall (E and I are Young's modulus of elasticity and moment of inertia, respectively). The summation of lateral stiffness of all columns and shear walls at a particular floor level constitute the total lateral stiffness of that particular story or floor level. The program checks soft story of a building along both global X and Z directions respectively. This computation is valid only for those structures whose floors are treated as rigid diaphragm.

**Related Links**

- **G.16.2 Seismic Load Generator** (on page 2343)
- **V. IS 1893 2002 Static Seismic** (on page 3483)

**TR.31.2.5.2 Identification of Floor Level**

The following two ways can identify floor level:

- Program calculated
- User defined
For regular building which has well defined floors (i.e., does not contain shear wall, staggered flooring, etc), STAAD can identify floor level on its own. However, if floor level is not so well defined, it is better to define floor height to have more accurate result for the purpose of torsion and soft story checking.

*Program calculated*

In general, STAAD identifies floor levels in order of increasing magnitude of Y-coordinates of joints. The program sorts different values of Y-coordinates, from minimum to maximum values, in ascending order and consider each Y-coordinate value as each floor level. This is the method used by the DEFINE UBC or similar load generation features.

This feature has been enhanced to identify the beam-column junctions at each floor level, as identified by the method above. If no beam-column junctions are identified at that level, a floor level will not be considered at that level in the structure. Where beam-column junctions are found, the program identifies two beams, at the same level which span in two different directions from the same beam-column junction. If this true, this identified floor level will be considered as truly existing floor level in the structure.

The enhanced feature for finding floor level is being used by lateral load generation feature for response spectrum analysis and soft story checking as per IS1893-2002 code. For response spectrum analysis, story drift and soft story checking is performed only when floor heights are specified.

*User defined*

Floor heights should be defined before using the DEFINE command for any primary response spectrum load case. The following general format is used for a user-defined floor height:

```
FLOOR HEIGHT

h_1 ; h_2 ; h_3 ; ... ; h_i - h_{i+1} ; ... ; h_n

(BASE h_b)
```

Where:

- \( h_1 \ldots h_n \) = the different floor heights in current length unit and \( n \) is the number of floor levels.
- \( h_b \) = the base level with respect to which 1st story height will be calculated. If \( h_b \) is not defined, the minimum Y-coordinate value present in the model will be taken as base level.

User defined floor heights are used by lateral load generation for response spectrum analysis and soft story checking as per IS1893-2002 code.

See TR.32.12.2 Generation of Seismic Loads (on page 2773) for an example of the correct usage of this command.

**TR.31.2.10 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load**

This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach per Part 1 (2016) for building structures.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads. This facility has not been developed for cases where the Z axis is set to be the vertical direction (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)).
General Format

```
DEFINE IS1893 2016 ( ACCIDENTAL ) LOAD

1893-2016-spec
WALL AREA { X | Z }
wall-data-pairs
SELFWEIGHT
JOINT WEIGHT
joint-list WEIGHT w
MEMBER WEIGHT

mem-list { UNT v1 v2 v3 | CON v4 v5 }
REFERENCE LOAD { X | Y | Z }

Ri1 f18
```

where

```
1893-2016-spec = ZONE f1 RF f2 I f3 { SS f4 | SA f11 } (ST f5) ( { DM f6 | DF f12 } ) (PX f7) (PZ f8) ( { DT f9 | GL f10 } ) (HT f13) (DX f14) (DZ f15)
```

**Note:** If both mass table data (SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT options) and a REFERENCE LOAD are specified, these will be added algebraically for a combined mass.

```
wall-data-pairs = w1, l1; w2, l2; w3, l3, ..., wn, ln;
```

where

- **ZONE f1** Seismic zone factor. Refer to Table 3 (Clause 6.4.2) of IS:1893 (Part 1)-2016.
- **RF f2** Response reduction factor. Refer to Table 9 (Clause 7.2.6) of IS: 1893 (Part 1) -2016.
- **I f3** Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance. Refer to Table 8 (Clause 7.2.3) of IS:1893 (Part 1)-2016.
- **SS f4** Rock or soil sites factor. Depending on type of soil, average response acceleration coefficient $S_a/g$ is calculated corresponding to 5% damping. Refer to Table 4 (Clause 6.4.2.1) of IS:1893 (Part 1) -2016.
  
  1 = hard soil  
  2 = medium soil  
  3 = soft soil

**Note:** Use either SS or SA to specify site conditions. If both parameters are specified, SS is ignored.

- **ST f5** The program will calculate natural period as per Clause 7.6.2 of IS:1893(Part 1)-2016.
  
  1 = RC moment-resisting frame building  
  2 = RC-Steel composite moment-resisting frame building  
  3 = Steel moment-resisting frame building
4 = Building with RC structural walls
5 = All other buildings

**DM**  <br> Damping ratio to obtain multiplying factor for calculating $S_a/g$ for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0. Refer to Clause 7.2.4 of IS:1893(Part 1)-2016.

**Note:** Use either DM or DF to specify damping. If both parameters are specified, DM is ignored.

This should be a value between 0 (zero) and 1.0, inclusive. For example, 7% damping should be specified as 0.07.

**PX**  <br> Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate $S_a/g$ for generation of seismic load along X direction.

**PZ**  <br> Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate $S_a/g$ for generation of seismic load along Z direction.

**DT**  <br> Depth of foundation below ground level. It should be defined in current units. If the depth of foundation is 30 m or more, the value of $A_h$ is taken as half the value obtained. If the foundation is placed between the ground level and 30 m depth, this value is linearly interpolated between $A_h$ and 0.5$A_h$. Refer to Clause 6.4.5 of IS:1893(Part 1)-2016.

**Note:** Use either DT or GL to specify foundation depth. If both parameters are specified, DT is ignored. See Node d below.

**GL**  <br> Y coordinate of ground level (or global Z coordinate for SET Z UP). A reduced lateral force is applied to levels below this height, per Clause 6.4.5.

**Note:** Use either DT or GL to specify foundation depth. If both parameters are specified, DT is ignored. See Node d below.

**SA**  <br> Average response spectral acceleration coefficient corresponding to site specific spectra. Refer to Clause 6.4.7 of IS:1893(Part 1)-2016.

**DF**  <br> Multiplying factor for calculating $S_a/g$.

**HT**  <br> Height of the building. Refer Clause 7.6.2 (a) and Fig. 5 of IS 1893 2016

**DX**  <br> Base dimension of the building in X direction at the plinth level. Refer Clause 7.6.2(b) or (c)

**DZ**  <br> Base dimension of the building in Z direction at the plinth level. Refer Clause 7.6.2(b) or (c)

**w1, l1; w2, l2; w3, l3, ...; wn, ln**  <br> Used to specify wall dimensions for calculating effective cross section area of wall in the first story of the building.

**UNI v1, v2, v3**  <br> Used when specifying a uniformly distributed load with a value of $v_1$ starting at a distance of $v_2$ from the start of the member and ending at a distance of $v_3$ from the start of the member. If $v_2$ and $v_3$ are omitted, the load is assumed to cover the entire length of the member.

**CON v4, v5**  <br> Used when specifying a concentrated force with a value of $v_4$ applied at a distance of $v_5$ from the start of the member. If $v_5$ is omitted, the load is assumed to act at the center of the member.

**Ri**  <br> Identification number of a previously defined reference load case. See **TR.31.6 Defining Reference Load Types** (on page 2642)
This is followed by the magnification factor, $f_{18}$ (required for reference loads). The default value is 1.0.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

**Notes**

a. If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the IS 1893 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands you have specified.

The ACC option along with accidental eccentricity factor (generally 0.05 as per IS 1893 code) needs to be provided in the 1893 seismic primary load case (i.e., 1893 LOAD X / Z f1 ACC f3 ). $f_2$ can be negative. See TR.32.12.2.2 Generation of IS:1893 Seismic Load (on page 2778)

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the ACCIDENTAL option should not be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the 1893 seismic primary load case (i.e., 1893 LOAD X / Z f1 DEC f2 ACC f3 ). For equivalent seismic analysis, $f_2$ is 1.5 and $f_3$ is 0.05 as per IS 1893 code. $f_1$ is always positive or zero, however $f_2$ can be negative. If $f_2$ is 0.0, only accidental torsion will be considered for this particular load case.

b. By default, STAAD.Pro calculates natural periods of the structure in both X and Z directions respectively which are used in calculation for base shear. If PX and PZ are included, the program will consider these values for calculation of average response acceleration coefficient. If ST is used instead of PX and PZ values, then the program will calculate natural period depending upon the empirical expression given in IS:1893 (Part 1)-2016.

c. In the case where no rigid floor diaphragm is present, STAAD.Pro identifies columns and shear walls (without openings) as vertical components for the purpose of computing lateral stiffness of the story.

The lateral stiffness of a column is calculated as:

$$\frac{12EI}{L^3}$$

where

- $E$ = Young's modulus
- $I$ = moment of inertia
- $L$ = length of the column

The lateral stiffness for a shear wall (without opening) is calculated as:

$$\frac{1}{\frac{12EI}{P_h^3} + \frac{1.2P_h}{AG}}$$

Which is the summation of inverse of flexural stiffness and inverse of shear stiffness, obtained as deflection of a cantilever wall under a single lateral load, $P$, at its top.

where

- $h$ = height
- $A$ = cross-sectional area
- $G$ = shear modulus of the wall

The summation of lateral stiffnesses of all columns and shear walls at a particular floor level constitutes the total lateral stiffness of that particular story or floor level. The program checks for a soft story of a building
along both global X and Z directions respectively. This computation is valid only for those structures whose floors are treated as rigid diaphragm.

d. Clause 6.4.5 of IS:1893 part-I -2016 stipulates that for underground structures and foundation at a depth 30m or below, the design horizontal spectrum ($A_h$ or $A_k$) value should be taken as half of the actual one for structures placed between ground level and 30 m depth the design horizontal acceleration spectrum must be interpolated between $A_h$ and ($0.5 \times A_h$). The reduction of $A_h$ should be done on the portion of the structure (mass situated below GL) located below ground.

![Diagram of base shear calculation above and below ground level](image)

**Figure 307: Base shear, VB, calculation above and below ground level, GL**

You can provide DT or GL parameter to tell the program what your actual depth of foundation below the ground level.

**Note:** The parameter DT should not be used to reduce $A_h$. Only GL should be used.

The program will then evaluate the multiplication factor on $A_h$ and calculate the base shear. This reduces the actual base shear for underground portion and the base shear, $V_B$ is distributed into story shear of that portion (for static analysis).

- For the portion of the structure above the ground, the design lateral force at the $i^{th}$ floor, $Q_i$:

$$Q_i = \left( \frac{W_i h_i^2}{\sum W_i h_i^2} \right) VB_s$$

where:

- $W_i$ = seismic weight of $i^{th}$ floor above the ground
- $h_i$ = height of $i^{th}$ floor above the ground
- $VB_s$ = horizontal base shear $= A_h \times W_s$
$W_s = $ seismic weight of the portion which is above the ground

- For the portion of the structure below the ground, the design lateral force at the $j^{th}$ floor, $Q_j$:

$$Q_j = \left( \frac{W_j h_j^2}{\sum W_j h_j^2} \right) V B_u$$

where

- $W_j = $ seismic weight of $j^{th}$ floor below the ground
- $h_j = $ height of $j^{th}$ floor below the ground
- $V B_u = $ horizontal base shear
- $W_u = $ seismic weight of the portion which is below the ground

**Methodology**

The design base shear is computed by STAAD.Pro for building structures as per IS: 1893 (Part 1) 2016:

$$V = A_h W$$

where

- $A_h = $ the design spectral acceleration based on Clause 6.4.2. This value is calculated for each mode.

$$= \frac{Z I S_a}{2 R g}$$

**Note:** All symbols and notations in the above equation are as per IS: 1893(Part 1) 2016.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads:

1. You provide seismic zone coefficient and desired 1893 specs through the DEFINE 1893 LOAD command.
2. The program calculates the structure period, $T$.
3. The program calculates $S_a/g$ utilizing $T$. For the $Y$ direction, $S_a/g = 2.5$ per clause 6.4.6.
4. The program calculates $V$ from the above equation. $W$ is obtained from mass table data entered via SELFWEIGHT, JOINT WEIGHT(s), MEMBER WEIGHT(S), and/or REFERENCE LOAD you provide through the DEFINE 1893 LOAD command.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893 procedures.

See TR.32.12 Generation of Loads (on page 2771) for additional information.

**IS 1893 2016 Implementation**

**Note:** STAAD.Pro can automatically comply with the reduced section property requirements of clause 6.4.3 for seismic loads with use of the MEMBER CRACKED CODE IS1893 2016 command. Refer to TR.20.10 Member Property Reduction Factors (on page 2485) for details.

7.2 Lateral Force - The design spectral acceleration, $A_h$, is based on Clause 6.4.2. This value is calculated for each mode and is multiplied by the seismic weight at each degree of freedom.

- 7.2.2 - A minimum of lateral base shear which needs to be distributed to each floor (or each node of each floor) in a building is calculated per Table 7 and Clause 7.2.2. This is used to determine a minimum base shear value.
- 7.2.3 - An additional importance factor has been added.
• 7.2.4 - 5% damping should be used for all structures, regardless of the material. The program will accept other values in the DM parameter, but a warning will be issued in the output.

• 7.2.6 - The user interface includes a list of response reduction factors taken from Table 10 of IS 1893 2016.

7.6 Equivalent Static Method

• 7.6.2 - Calculation of the approximate time-period based on height of the building

  For point a, the time period is calculated as follows for reinforced-concrete and steel composite MRF builds: \( T_a = 0.080.75 \).

  For point b and point c, the time period is a function of the wall area. Therefore, the wall-data-pairs must provided to correctly calculate the time period.

Related Links
TR.31.2.14 IBC 2015 Seismic Load Definition
• TR.31.2.14 IBC 2015 Seismic Load Definition (on page 2600)
• V. IS 1893 2016 Static Seismic (on page 3492)

TR.31.2.11 IBC 2000/2003 Load Definition

The specifications of the IBC 2000, and 2003 codes for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

General Format

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE IBC 2000 or 2003 LOAD command.

```
DEFINE IBC ( { 2000 | 2003 } ) (ACCIDENTAL) LOAD
  ibc-spec
  SELFWEIGHT
  JOINT WEIGHT
  joint-list
  WEIGHT w
  MEMBER WEIGHT
  mem-list { UNI v1 v2 v3 | CON
  ELEMENT WEIGHT
  plate-list PRESS p1
  FLOOR WEIGHT
  YRANGE ...
```

Note: See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

```
ibc-spec = { SDS f1 SD1 f2 S1 f3 I f4 RX f5 RZ f6 SCLASS f7 (CT f8) (PX f9) (PZ f10) }

SDS f1  Design spectral response acceleration at short periods. See equation 16-18, Section 1615.1.3 of IBC 2000 and equation 9.4.1.2.5-1 of ASCE7-02

SD1 f2  Design spectral response acceleration at 1-second period. See equation 16-19, Section 1615.1.3. of IBC 2000 and equation 9.4.1.2.5-2 of ASCE7-02
```
S1 \textit{f}3  \hspace{1cm} \text{Mapped spectral acceleration for a 1-second period. See equation 16-17 of IBC 2000, and 9.4.1.2.4-2 of ASCE 7-02}

I \textit{f}4  \hspace{1cm} \text{Occupancy importance factor determined in accordance with Section 1616.2 of IBC 2000 and 2003, and section 9.1.4 (page 96) of ASCE 7-02}

RX \textit{f}5  \hspace{1cm} \text{The response modification factor for lateral load along the X direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table 1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000}

RZ \textit{f}6  \hspace{1cm} \text{The response modification factor for lateral load along the Z direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table 1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000.}

SCLASS \textit{f}7  \hspace{1cm} \text{Site class as defined in Section 1615.1.1 of IBC 2000 (page 350) & 2003 (page 322).}

<table>
<thead>
<tr>
<th>STAAD Value</th>
<th>IBC Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>A</td>
</tr>
<tr>
<td>2</td>
<td>B</td>
</tr>
<tr>
<td>3</td>
<td>C</td>
</tr>
<tr>
<td>4</td>
<td>D</td>
</tr>
<tr>
<td>5</td>
<td>E</td>
</tr>
<tr>
<td>6</td>
<td>F</td>
</tr>
</tbody>
</table>

CT \textit{f}8  \hspace{1cm} \text{Optional } C_t \text{ value used to calculate time period, } T_a. \text{ See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02. If specified, it is your responsibility to provide the value in the correct system of units. Refer to Table 9.5.5.3.2 of AISC 7-02 for values.}

If the value of } C_t \text{ is not provided, then the program computes the average value of the modulus of elasticity of the model, } E_{avg} = \sum E / M \text{ (where } M \text{ is the number of members) and uses this to determine the structure type:}

i. } E_{avg} < 4,000 \text{ ksi, the program uses a } C_t \text{ for a moment-resisting concrete frame.}

ii. } E_{avg} > 10,000 \text{ ksi, the program uses a } C_t \text{ for a moment-resisting steel frame.}

iii. } 4,000 \text{ ksi} \leq E_{avg} \leq 10,000 \text{ ksi, the program uses a } C_t \text{ value for "all other structural systems".}

**Note:** It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when } C_t \text{ not specified. Refer to the IBC/ASCE 7 code for detailed descriptions.}

Table 9.5.5.3.2 of ASCE 7-02 also includes “Eccentrically braced steel frames”. STAAD.Pro does not select this value automatically. For this structure type, you must specify } C_t.

PX \textit{f}9  \hspace{1cm} \text{Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.
Optional Period of structure (in sec) in Z or Y direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The implementation details of the respective codes are as follows:

IBC 2000

On a broad basis, the rules described in section 1617.4 of the IBC 2000 code document have been implemented. These are described in pages 359 thru 362 of that document. The specific section numbers, those which are implemented, and those which are not implemented, are as follows:

Table 242: Sections of IBC 2000 implemented and omitted in the program

<table>
<thead>
<tr>
<th>Implemented sections of IBC 2000</th>
<th>Omitted sections of IBC 2000</th>
</tr>
</thead>
<tbody>
<tr>
<td>1617.4.1</td>
<td>1617.4.4.1</td>
</tr>
<tr>
<td>1617.4.1.1</td>
<td>1617.4.4.2</td>
</tr>
<tr>
<td>1617.4.2</td>
<td>1617.4.4.3</td>
</tr>
<tr>
<td>1617.4.2.1</td>
<td>1617.4.4.5</td>
</tr>
<tr>
<td>1617.4.3</td>
<td>1617.4.5</td>
</tr>
<tr>
<td>1617.4.4</td>
<td>1617.4.6</td>
</tr>
<tr>
<td>1617.4.4.4</td>
<td></td>
</tr>
</tbody>
</table>

IBC 2003

On a broad basis, the rules described in section 1617.4 of the IBC 2003 code document have been implemented. This section directs the engineer to Section 9.5.5 of the ASCE 7 code. The specific section numbers of ASCE 7-2002, those which are implemented, and those which are not implemented, are shown in the table below. The associated pages of the ASCE 7-2002 code are 146 thru 149.
Table 243: Sections of IBC 2003 (ASCE 7-02) implemented and omitted in the program

<table>
<thead>
<tr>
<th>Implemented sections of IBC 2003 (ASCE 7-02)</th>
<th>Omitted sections of IBC 2003 (ASCE 7-02)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.5.5.2</td>
<td>9.5.5.5.1</td>
</tr>
<tr>
<td>9.5.5.2.1</td>
<td>9.5.5.5.2</td>
</tr>
<tr>
<td>9.5.5.3</td>
<td>9.5.5.6</td>
</tr>
<tr>
<td>9.5.5.3.1</td>
<td>9.5.5.7</td>
</tr>
<tr>
<td>9.5.5.3.2</td>
<td></td>
</tr>
<tr>
<td>9.5.5.4</td>
<td></td>
</tr>
<tr>
<td>9.5.5.5</td>
<td></td>
</tr>
<tr>
<td>Portions of 9.5.5.5.2</td>
<td></td>
</tr>
</tbody>
</table>

Methodology

The design base shear is computed in accordance with Eqn. 16-34 of IBC 2000 and Eqn. 9.5.5.2-1 of ASCE 7-02:

$$V = C_s W$$

The seismic response coefficient, $C_s$, is determined in accordance with Eqn. 16-35 of IBC 2000 / Eqn. 9.5.5.2.1-1 of ASCE 7-02:

$$C_s = \frac{S_{DS}}{R/I_E}$$

$C_s$ need not exceed the limit given in Eqn. 16-36 of IBC 2000 / Eqn. 9.5.5.2.1-2 of ASCE 7-02:

$$C_s = \frac{S_{D1}}{(R/I_E)T}$$

$C_s$ shall not be taken less than the lower limit given in Eqn. 16-37 of IBC 2000 / Eqn. 9.5.5.2.1-3 of ASCE 7-0:

$$C_s = 0.044 S_{DS} I_E$$

In addition, for structures for which the 1-second spectral response, $S_1$, is equal to or greater than 0.6g, the value of the seismic response coefficient, $C_s$, shall not be taken less than the limit given in Eqn. 16-38 of IBC 2000 / Eqn. 9.5.5.2.1-4 of ASCE 7-02:

$$C_s = \frac{0.5 S_1}{R/I_E}$$

For an explanation of the terms used in the above equations, please refer to the relevant IBC and ASCE 7-02 codes.

Procedure Used by the Program

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02 (IBC 2003) This is reported in the output as $T_a$.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as $T$. 
3. you may override the Rayleigh based period by specifying a value for \( PX \) or \( PZ \) depending on the direction of the IBC load.

4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in clause 1617.4.2 of IBC 2000, section 9.5.5.3 of ASCE 7-02 (IBC 2003) or section 12.8.2.1 of ASCE 7-05 (IBC 2006). The resulting value is reported as “Time Period used.”

5. The Design Base Shear is calculated based on equation 16-34 of IBC 2000, equation 9.5.5.2-1 of ASCE 7-02 (IBC 2003) or equation 12.8-1 of ASCE 7-05 (IBC 2006). It is then distributed at each floor using the rules of clause 1617.4.3, equations 16-41 and 16-42 of IBC 2000. For IBC 2003, using clause 9.5.5.4, equations 9.5.5.4-1 & 9.5.5.4-2 of ASCE 7-02.

6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (clause 1617.4.4.4 of IBC 2000, and section 9.5.5.5.2 of ASCE 7-02 for IBC 2003). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.

Example 1

```
DEFINE IBC 2003 LOAD
SDS 0.6 SD1 0.36 S1 0.31 I 1.0 RX 3 RZ 4 SCL 4 CT 0.032
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2771) for details.

```
LOAD 1 ( Seismic Load in X Direction )
IBC LOAD X 0.75
LOAD 2 ( Seismic Load in Z Direction )
IBC LOAD Z 0.75
```

The Examples manual contains examples illustrating load generation involving IBC and UBC load types.

Related Links
- G.16.2 Seismic Load Generator (on page 2343)

**TR.31.2.12 IBC 2006/2009 Seismic Load Definition**

The specifications of the seismic loading chapters of the International Code Council 2006 & 2009 code and the ASCE 7-05 (including Supplement #2) code for seismic analysis of a building using a static equivalent approach are available in the program. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).
General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE IBC 2006 LOAD command.

**DEFINE IBC 2006 (ACCIDENTAL) LOAD**

*map-spec* *ibc06-spec*

**SELFWEIGHT**
**JOINT WEIGHT**
**joint-list**
**WEIGHT w**
**MEMBER WEIGHT**
**mem-list**

**ELEMENT WEIGHT**
**plate-list**
**PRESS p**
**FLOOR WEIGHT**
**YRANGE** …

**Note:** See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

*map-spec* = \{ ZIP \{ f_1 \} | LAT \{ f_2 \} LONG \{ f_3 \} | SS \{ f_4 \} S1 \{ f_5 \} \}

Where:

- **ZIP** \( f_1 \): The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-05 Chapter 22).
- **LAT** \( f_2 \): The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-05 Chapter 22).
- **LONG** \( f_3 \): The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-05 Chapter 22).
- **SS** \( f_4 \): The mapped MCE for 0.2s spectral response acceleration. (IBC 2006/2009 Clause 1613.5.1, ASCE 7-05 Clause 11.4.1).
- **S1** \( f_5 \): The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-05

*ibc06-spec* = \{ RX \{ f_6 \} RZ \{ f_7 \} I \{ f_8 \} TL \{ f_9 \} SCLASS \{ f_{10} \} (CT \{ f_{11} \} (PX \{ f_{12} \} (PZ \{ f_{13} \} (K \{ f_{14} \} (FA \{ f_{15} \} (EV \{ f_{16} \} ) \)

Where:

- **RX** \( f_6 \): The response modification factor, R, for lateral load along the X direction, (ASCE Table 12.2.1). This is the value used for calculating \( C_x \).
- **RZ** \( f_7 \): The response modification factor, R, for lateral load along the Z direction, (ASCE Table 12.2.1). This is the value used for calculating \( C_y \).
- **I** \( f_8 \): Occupancy importance factor. (IBC 2006/2009 Clause 1604.5, ASCE 7-05 Table 11.5-1)
- **TL** \( f_9 \): Long-Period transition period in seconds. (ASCE 7-05 Clause 11.4.5 and Chapter 22).
- **SCLASS** \( f_{10} \): Site class. Enter 1 through 6 in place of A through F, see table below. (IBC 2006/2009 Table 1613.5.2, ASCE 7-05 Section 20.3)
Optional $C_t$ value in X-direction to calculate time period. (ASCE 7-05 Table 12.8-2). If specified, it is your responsibility to provide the value in the correct system of units. Refer to AISC 7-05 for values.

If the value of $C_t$ is not provided, then the program computes the average value of the modulus of elasticity of the model, $E_{avg} = \sum E / M$ (where $M$ is the number of members) and uses this to determine the structure type:

i. $E_{avg} < 4,000$ ksi, the program uses a $C_t$ for a moment-resisting concrete frame.
ii. $E_{avg} > 10,000$ ksi, the program uses a $C_t$ for a moment-resisting steel frame.
iii. $4,000$ ksi $\leq E_{avg} \leq 10,000$ ksi, the program uses a $C_t$ value for “all other structural systems”.

Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when $C_t$ not specified. Refer to the IBC/ASCE 7 code for detailed descriptions.

ASCE 7-05 also includes “Eccentrically braced steel frames”. STAAD.Pro does not select this value automatically. For this structure type, you must specify $C_t$.

Optional $C_t$ value in Z-direction to calculate time period. (ASCE 7-05 Table 12.8-2). Refer to CTX for details.

Optional period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-05 Table 12.8-2).

Optional period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-05 Table 12.8-2).

Optional exponent value, $x$, in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-05 table 12.8-2 p129). If the value of $x$ is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

Optional exponent value, $x$, in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-05 table 12.8-2 p129). If the value of $x$ is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006/2009 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).

Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006/2009 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

Implementation in STAAD.Pro

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The rules described in section 1613 of the ICC IBC-2006 code (except 1613.5.5) have been implemented. This section directs the engineer to the ASCE 7-2005 code. The specific section numbers of ASCE 7— those which are implemented, and those which are not implemented—are shown in the table below.
### Table 245: Sections of IBC 2006 implemented and omitted in the program

<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>11.4</td>
<td>12.8.4.1</td>
</tr>
<tr>
<td>11.5</td>
<td>12.8.4.3 and onwards</td>
</tr>
<tr>
<td>12.8</td>
<td></td>
</tr>
</tbody>
</table>

Additionally, Supplement #2 of ASCE 7-05— as referenced by IBC 2009— specifies a different equation to be used for the lower bound of the seismic response coefficient, which has also been implemented.

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 12.8.2.1 of ASCE 7-05 (IBC 2006/2009). This is reported in the output as $T_a$.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as $T$.
3. You may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in section 12.8.2 of ASCE 7-05 (IBC 2006). The resulting value is reported as “Time Period used” in the output file.
5. The Design Base Shear is calculated based on equation 12.8-1 of ASCE 7-05 (IBC 2006). It is then distributed at each floor using the rules of clause 12.83, equations 12.8-11, 12.8-12 and 12.8-13 of ASCE 7-05.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (section 12.8.4.2 of ASCE 7-05 for IBC 2006). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. The amplification of accidental torsional moment, as described in Section 12.8.4.3 of the ASCE 7-05 code, is not implemented.
8. The story drift determination as explained in Section 12.8.6 of the ASCE 7-05 code is not implemented in STAAD.Pro.

### Methodology

The design base shear is computed in accordance with the following equation (equation 12.8-1 of ASCE 7-05):

$$ V = C_s W $$

The seismic response coefficient, $C_s$, is determined in accordance with the following equation (equation 12.8-2 of ASCE 7-05):

$$ C_s = \frac{S_{DS}}{[R/IE]} $$

For IBC 2006, $C_s$ need not exceed the following limits defined in ASCE 7-05 (equations 12.8-3 and 12.8-4):

- $C_s = \frac{S_{DL}}{[T \cdot (R/IE)]}$ for $T \leq T_L$
- $C_s = \frac{S_{DL} \cdot T_L}{[T^o \cdot (R/IE)]}$ for $T > T_L$

However, $C_s$ shall not be less than (equation 12.8-5 of ASCE 7-05, supplement #2):
\[ C_s = 0.044 \cdot S_{DS} \cdot I \geq 0.01 \]

In addition, per equation 12.8-6 of ASCE 7-05, for structures located where \( S_1 \) is equal to or greater than 0.6g, \( C_s \) shall not be less than

\[ C_s = 0.5 \cdot S_1/(R/I) \]

For an explanation of the terms used in the above equations, please refer to the IBC 2006/2009 and ASCE 7-05 codes.

**Example 1**
```
DEFINE IBC 2006
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

**Example 2**
The following example shows the commands required to enable the program to generate the lateral loads. Refer to [TR.32.12 Generation of Loads](on page 2771) of the Technical Reference Manual for this information.
```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

**Related Links**
- [G.16.2 Seismic Load Generator](on page 2343)

**TR.31.2.13 IBC 2012 Seismic Load Definition**
The specifications of the seismic loading chapters of the International Code Council 2012 code and the ASCE 7-10 code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

**General Format**
There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE IBC 2012 LOAD` command.
```
DEFINE IBC 2012 (ACCIDENTAL) LOAD
map-spec ibc12-spec
SELFWEIGHT
JOINT WEIGHT
joint-list
WEIGHT w
MEMBER WEIGHT
```
where:

**map-spec** = { **ZIP** f1 | **LAT** f2 **LONG** f3 | **SS** f4 **S1** f5 }

Where:

**ZIP** f1  The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-10 Chapter 22).

**LAT** f2  The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).

**LONG** f3  The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).

**SS** f4  The mapped MCE for 0.2s spectral response acceleration. (IBC 2012 Clause 1613.5.1, ASCE 7-10 Clause 11.4.1).

**S1** f5  The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-10.

**ibc12-spec** = { **RX** f6 **RZ** f7 **I** f8 **TL** f9 **SCLASS** f10 (**CTX** f11) (**CTZ** f12) (**PX** f13) (**PZ** f14) (**XX** f15) (**XZ** f16) (**FA** f17) (**FV** f18) }

Where:

**RX** f6  The response modification factor, R, for lateral load along the X direction, (ASCE 7-10 Table 12.2.1). This is the value used for calculating $C_s$.

**RZ** f7  The response modification factor, R, for lateral load along the Z direction, (ASCE 7-10 Table 12.2.1) This is the value used for calculating $C_s$.

**I** f8  Occupancy importance factor (IBC 2012 Clause 1604.5, ASCE 7-10 Table 11.5-1).

**TL** f9  Long-Period transition period in seconds (ASCE 7-10 Clause 11.4.5 and Chapter 22).

**SCLASS** f10  Site class. Enter 1 through 6 in place of A through F, see table below (IBC 2012 clause 1613.3.2, ASCE 7-10 Section 20.3)

**CTX** f11  Optional $C_t$ value in X-direction to calculate time period. (ASCE 7-10 Table 12.8-2). If specified, it is your responsibility to provide the value in the correct system of units. Refer to AISC 7-10 for values.

If the value of $C_t$ is not provided, then the program computes the average value of the modulus of elasticity of the model, $E_{avg} = \sum E / M$ (where $M$ is the number of members) and uses this to determine the structure type:

- i. $E_{avg} < 4,000$ ksi, the program uses a $C_t$ for a moment-resisting concrete frame.
- ii. $E_{avg} > 10,000$ ksi, the program uses a $C_t$ for a moment-resisting steel frame.
- iii. $4,000$ ksi $\leq E_{avg} \leq 10,000$ ksi, the program uses a $C_t$ value for “all other structural systems”.

### Note:
See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.
Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when \( C_t \) not specified. Refer to the IBC/ASCE 7 code for detailed descriptions.

ASCE 7-10 also includes "Eccentrically braced steel frames". STAAD.Pro does not select this value automatically. For this structure type, you must specify \( C_t \).

CTZ \( f_{12} \) Optional \( C_t \) value in Z-direction to calculate time period. (ASCE 7-10 Table 12.8-2). Refer to CTX for details.

PX \( f_{13} \) Optional period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

PZ \( f_{14} \) Optional period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

XX \( f_{15} \) Optional exponent value, \( x \), in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of \( x \) is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

XZ \( f_{16} \) Optional exponent value, \( x \), in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of \( x \) is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

FA \( f_{17} \) Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2012 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

FV \( f_{18} \) Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2012 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

Implementation in STAAD.Pro

Note: This feature requires STAAD.Pro 2007 (Build 11) or higher. Refer to AD.2007-11.3.7 of the What’s New section for additional information on using this feature.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The rules described in section 1613 of the ICC IBC-2012 code (except 1613.5.5) have been implemented. This section directs the engineer to the ASCE 7-2010 code. The specific section numbers of ASCE 7—those which are implemented, and those which are not implemented—are shown in the table below.
Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 12.8.2.1 of ASCE 7-10 (IBC 2012). This is reported in the output as Ta.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T.
3. You may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in section 12.8.2 of ASCE 7-10 (IBC 2012). The resulting value is reported as "Time Period used" in the output file.
5. The Design Base Shear is calculated based on equation 12.8-1 of ASCE 7-10 (IBC 2012). It is then distributed at each floor using the rules of clause 12.8.3, equations 12.8-11, 12.8-12 and 12.8-13 of ASCE 7-10.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (section 12.8.4.2 of ASCE 7-10 for IBC 2012). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. The amplification of accidental torsional moment, as described in Section 12.8.4.3 of the ASCE 7-10 code, is not implemented.
8. The story drift determination as explained in Section 12.8.6 of the ASCE 7-10 code is not implemented in STAAD.

Methodology

The design base shear is computed in accordance with the following equation (equation 12.8-1 of ASCE 7-10):

\[ V = C_s W \]

The seismic response coefficient, \( C_s \), is determined in accordance with the following equation (equation 12.8-2 of ASCE 7-10):

\[ C_s = S_{DS}/[R/I_E] \]

For IBC 2012, \( C_s \) need not exceed the following limits defined in ASCE 7-10 (equations 12.8-3 and 12.8-4):

\[ C_s = S_{D1}/[T \cdot (R/I)] \] for \( T \leq T_L \)
\[ C_s = S_{D1} \cdot T_L/[T^2(R/I)] \] for \( T > T_L \)

However, \( C_s \) shall not be less than (equation 12.8-5 of ASCE 7-10):

\[ C_s = 0.044 \cdot S_{DS} \cdot I \geq 0.01 \]
In addition, per equation 12.8-6 of ASCE 7-10, for structures located where \( S_1 \) is equal to or greater than 0.6g, \( C_s \) shall not be less than

\[
C_s = 0.5 \cdot \frac{S_1}{(R/I)}
\]

For an explanation of the terms used in the above equations, please refer to the IBC 2012 and ASCE 7-10 codes.

**Example 1**

```
DEFINE IBC 2012
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

**Example 2**

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2771) for this information.

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

**Related Links**

- [V. IBC 2012 Static Seismic](on page 3427)

**TR.31.2.14 IBC 2015 Seismic Load Definition**

The specifications of the seismic loading chapters of the International Code Council 2015 code and the ASCE 7-10 code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

**General Format**

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE IBC 2015 LOAD` command.

```
DEFINE IBC 2015 (ACCIDENTAL) LOAD
map-spec ibc15-spec

SELFWEIGHT
JOINT WEIGHT
joint-list
WEIGHT w
MEMBER WEIGHT
mem-list { UNI v1 v2 v3 | CON
ELEMENT WEIGHT
plate-list PRESS p1
FLOOR WEIGHT
YRANGE ...
```
Note: See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

\[ \text{map-spec} = \{ \text{ZIP } f_1 \mid \text{LAT } f_2 \mid \text{LONG } f_3 \mid \text{SS } f_4 \mid \text{S1 } f_5 \} \]

Where:

- **ZIP \( f_1 \)**: The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-10 Chapter 22).
- **LAT \( f_2 \)**: The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).
- **LONG \( f_3 \)**: The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).
- **SS \( f_4 \)**: The mapped MCE for 0.2s spectral response acceleration. (IBC 2015 Clause 1613.5.1, ASCE 7-10 Clause 11.4.1).
- **S1 \( f_5 \)**: The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-10.

\[ \text{ibc15-spec} = \{ \text{RX } f_6 \mid \text{RZ } f_7 \mid \text{I } f_8 \mid \text{TL } f_9 \mid \text{SCLASS } f_{10} \mid \text{CTX } f_{11} \mid \text{CTZ } f_{12} \mid \text{PX } f_{13} \mid \text{PZ } f_{14} \mid \text{XX } f_{15} \mid \text{XZ } f_{16} \mid \text{FA } f_{17} \mid \text{FV } f_{18} \} \]

Where:

- **RX \( f_6 \)**: The response modification factor, R, for lateral load along the X direction, (ASCE 7-10 Table 12.2.1). This is the value used for calculating \( C_s \).
- **RZ \( f_7 \)**: The response modification factor, R, for lateral load along the Z direction, (ASCE 7-10 Table 12.2.1). This is the value used for calculating \( C_s \).
- **I \( f_8 \)**: Occupancy importance factor (IBC 2015 Clause 1604.5, ASCE 7-10 Table 11.5-1).
- **TL \( f_9 \)**: Long-Period transition period in seconds (ASCE 7-10 Clause 11.4.5 and Chapter 22).
- **SCLASS \( f_{10} \)**: Site class. Enter 1 through 6 in place of A through F, see table below (IBC 2015 clause 1613.3.2, ASCE 7-10 Section 20.3).
- **CTX \( f_{11} \)**: Optional CT value in X-direction to calculate time period. (ASCE 7-10 Table 12.8-2). If specified, it is your responsibility to provide the value in the correct system of units. Refer to AISC 7-10 for values.

If the value of \( C_t \) is not provided, then the program computes the average value of the modulus of elasticity of the model, \( E_{avg} = \sum E / M \) (where \( M \) is the number of members) and uses this to determine the structure type:

1. \( E_{avg} < 4,000 \text{ ksi} \), the program uses a \( C_t \) for a moment-resisting concrete frame.
2. \( E_{avg} > 10,000 \text{ ksi} \), the program uses a \( C_t \) for a moment-resisting steel frame.
3. \( 4,000 \text{ ksi} \leq E_{avg} \leq 10,000 \text{ ksi} \), the program uses a \( C_t \) value for “all other structural systems”.

Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when \( C_t \) not specified. Refer to the IBC/ASCE 7 code for detailed descriptions.
ASCE 7-10 also includes “Eccentrically braced steel frames”. STAAD.Pro does not select this value automatically. For this structure type, you must specify \( C_t \).

**CTZ** 
Optional CT value in Z-direction to calculate time period. (ASCE 7-10 Table 12.8-2). Refer to CTX for details.

**PX** 
Optional period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

**PZ** 
Optional period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

**XX** 
Optional exponent value, \( x \), in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of \( x \) is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

**XZ** 
Optional exponent value, \( x \), in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of \( x \) is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

**FA** 
Optional Short-Period site coefficient at 0.2s. Value must be provided if \( SCLASS \) set to F (i.e., 6). (IBC 2015 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

**FV** 
Optional Long-Period site coefficient at 1.0s. Value must be provided if \( SCLASS \) set to F (i.e., 6). (IBC 2015 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

### Implementation and Methodology

Refer to TR.31.2.13 IBC 2012 Seismic Load Definition (on page 2596) for details on the implementation of the IBC 2015 / IBC 2012 / ASCE 7-10 static seismic method.

#### Example 1

```plaintext
DEFINE IBC 2015
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

#### Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2771) for this information.

```plaintext
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```
Related Links

- *V. IBC 2015 Static Seismic* (on page 3436)

**TR.31.2.15 IBC 2018 Seismic Load Definition**

The specifications of the seismic loading chapters of the International Code Council 2018 code and the ASCE 7-16 code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

**General Format**

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE IBC 2018 LOAD` command.

```
DEFINE IBC 2018 (ACCIDENTAL) LOAD
map-spec  ibc18-spec

SELFWEIGHT
JOINT WEIGHT
joint-list
WEIGHT w
MEMBER WEIGHT
mem-list { UNI v1 v2 v3 | CON
ELEMENT WEIGHT
plate-list PRESS p1
FLOOR WEIGHT
YRANGE ...
```

**Note:** See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for `FLOOR WEIGHT` input description.

Where:

```
map-spec = { ZIP f1 | LAT f2 LONG f3 | SS f4 S1 f5 }
```

Where:

- **ZIP f1** The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-16 Chapter 22).
- **LAT f2** The latitude of the site used to determine the Ss and S1 factors. (ASCE 7-16 Chapter 22).
- **LONG f3** The longitude of the site used to determine the Ss and S1 factors. (ASCE 7-16 Chapter 22).
- **SS f4** The mapped MCE for 0.2s spectral response acceleration. (ASCE 7-16 Clause 11.4.2).
- **S1 f5** The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.2 ASCE7-16.

```
ibc18-spec = { RX f6 RZ f7 I f8 TL f9 SCLASS f10 (CTX f11) (CTZ f12) (PX f13) (PZ f14) (XX f15) (XZ f16) (FA f17) (FV f18) }
```

Where:

- **RX f6** The response modification factor, R, for lateral load along the X direction, (ASCE Table 12.2.1). This is the value used for calculating $C_x$. 
The response modification factor, \( R \), for lateral load along the Z direction, (ASCE Table 12.2.1)
This is the value used for calculating \( C_s \).

Occupancy importance factor. (IBC 2018 Clause 1604.5, ASCE 7-16 Table 11.5-1)

Long-Period transition period in seconds. (ASCE 7-16 Clause 11.4.5 and Chapter 22).

Site class. Enter 1 through 6 in place of A through F, see table below. (IBC 2018 clause 1613.3.2, ASCE 7-16 Section 20.3)

Optional CT value in X-direction to calculate time period. (ASCE 7-16 Table 12.8-2). If specified, it
is your responsibility to provide the value in the correct system of units. Refer to AISI 7-16 for
values.

If the value of \( C_t \) is not provided, then the program computes the average value of the modulus of
elasticity of the model, \( E_{avg} = \sum E / M \) (where \( M \) is the number of members) and uses this to
determine the structure type:

i. \( E_{avg} < 4,000 \text{ ksi} \), the program uses a \( C_t \) for a moment-resisting concrete frame.

ii. \( E_{avg} > 10,000 \text{ ksi} \), the program uses a \( C_t \) for a moment-resisting steel frame.

iii. \( 4,000 \text{ ksi} \leq E_{avg} \leq 10,000 \text{ ksi} \), the program uses a \( C_t \) value for “all other structural systems”.

Note: It is your responsibility to ensure that the structure type used actually matches the
description for the automatically determined structure when \( C_t \) not specified. Refer to the IBC/
ASCE 7 code for detailed descriptions.

ASCE 7-16 also includes “Eccentrically braced steel frames”. STAAD.Pro does not select this value
automatically. For this structure type, you must specify \( C_t \).

Optional CT value in Z-direction to calculate time period. (ASCE 7-16 Table 12.8-2).

Refer to CTX for details.

Optional period of structure (in sec) in X-direction to be used as fundamental period of the
structure. If not entered the value is calculated from the code. (ASCE 7-15 Table 12.8-2).

Optional period of structure (in sec) in Z-direction to be used as fundamental period of the
structure. If not entered the value is calculated from the code. (ASCE 7-16 Table 12.8-2).

Optional exponent value, \( x \), in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-16 table
12.8-2). If the value of \( x \) is not provided, then the program computes the average value of the
modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

Optional exponent value, \( x \), in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-16 table
12.8-2). If the value of \( x \) is not provided, then the program computes the average value of the
modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6).
(IBC 2018 Clause 1613.3.3, ASCE 7-16 Section 11.4.3).

Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6).
(IBC 2018 Clause 1613.3.3, ASCE 7-16 Section 11.4.3).

Implementation and Methodology

Refer to TR.31.2.13 IBC 2012 Seismic Load Definition (on page 2596) for details on the implementation of the
Example 1

DEF IBC 2018
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
s1 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2771) for this information.

LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75

Related Links

- V.IBC 2018 Static Seismic T 1.2 (on page 3440)
- V.IBC 2018 Static Seismic T Greater Than 2.5 (on page 3450)
- V.IBC 2018 Static Seismic T Less Than 0.5 (on page 3468)

TR.31.2.16 Japanese Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per Japanese specifications using a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). The implementation is as per Article 88 in the “Building Codes Enforcement Ordinance 2006”.

General Format

DEFINE AIJ (ACCIDENTAL) LOAD

AIJ-spec

SELFWEIGHT
JOINT WEIGHT
joint-list
WEIGHT w
MEMBER WEIGHT
mem-list { UNI v1 v2 v3 | CON
ELEMEIN WEIGHT
plate-list PRESS p1
FLOOR WEIGHT
YRANGE ...

Note: See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.
Where:

\[ \text{AIJ-spec} = \{ \text{ZONE } f_1 \text{ CO } f_2 \text{ TC } f_3 \text{ ALPHA } f_4 \} \]

Where:

- **ZONE \( f_1 \)**: Zone factor (0.7, 0.8, 0.9 or 1.0)
- **CO \( f_2 \)**: Normal coefficient of shear force (0.2 or 1.0)
- **TC \( f_3 \)**: Period defined by ground support specification (0.4, 0.6 or 0.8 sec)
- **ALPHA \( f_4 \)**: Ratio of steel height to overall building height, which are used in the calculation of Rt.

### Generation of AIJ Seismic Load

General format to provide Japanese Seismic load in a primary load case:

```plaintext
LOAD i
AIJ LOAD {X | Y | Z} (f5) (ACC f6)
```

Where:

- **LOAD \( i \)**: load case number
- **AIJ LOAD \{ X | Y | Z \} \( f_5 \)**: optional factor to multiply horizontal seismic load.
- **ACC \( f_6 \)**: multiplying factor for Accidental Torsion, to be used to multiply the AIJ accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

**Note:** Choose horizontal directions only.

If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the AIJ specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to [TR.32.12.2.1 Generation of Seismic Loads](on page 2773).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in [TR.5 Set Command Specification](on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

### Methodology

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE AIJ LOAD command.

Program calculates the natural period of building \( T \) utilizing the following equation:

\[ T = h(0.02 + 0.01\alpha) \]

where

- **\( h \)**: height of building (m)
- **\( \alpha \)**: Alpha, ratio of steel height for overall height
Design spectral coefficient, $R_t$, is calculated utilizing $T$ and $T_c$ as follows:

$$R_t = \begin{cases} 1.0 & \text{when } T < T_c \\ 1 - 0.2(T / T_c - 1)^2 & \text{when } T_c \leq T \leq 2T_c \\ 1.6T_c / T & \text{when } 2T_c < T \end{cases}$$

where

$$T_c = \text{Period defined by ground support specification for } R_t.$$

$\alpha_i$ is calculated from the weight values provided in the Define AIJ Load command.

$$\alpha_i = W_i / W$$

where

$$W_i = \text{sum of weight from a top to floor } i.$$ $W = \text{all weight}$

The seismic coefficient of floor, $C_i$, is calculated using:

$$C_i = Z R_t A_i C_o$$

where

$$Z = \text{zone factor}$$

$$C_o = \text{normal coefficient of shear force}$$

$$A_i = 1 + \frac{2T(1/\sqrt{a_i} - a_i)}{1 + 3T}$$

Seismic load shear force, $Q_i$, of each floor is calculated by $C_i$ and $W_i$:

$$Q_i = C_i W_i$$

where

$$W_i = \text{sum of weight from a top to i floor}$$

Load value of each floor $P_i$ is calculated by seismic load shear force $Q_i$.

$$P_i = Q_i - Q_{i+1}$$

The total lateral seismic load is distributed by the program among different levels.

### Example

```plaintext
DEFINE AIJ LOAD
ZONE 0.8 CO 0.2 TC 0.6 alpha 1.0
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X)
AIJ LOAD X
```

### Related Links
- [G.16.2 Seismic Load Generator](on page 2343)

### TR.31.2.17 CFE (Comisión Federal De Electricidad) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per MANUAL DE DISEÑO POR SISMO - SEISMIC DESIGN HANDBOOK COMISIÓN FEDERAL DE ELECTRICIDAD - ELECTRIC
POWER FEDERAL COMMISSION - October 1993 (Chapters 3.1, 3.2, 3.3 and 3.4) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). This is a code used in the country of Mexico.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

**General Format**

```
DEFINE CFE (ACCIDENTAL) LOAD
cfe-spec
  SELFWEIGHT
  JOINT WEIGHT
  joint-list
  WEIGHT w
  MEMBER WEIGHT
  mem-list { UNI v1 v2 v3 | CON
  ELEMENT WEIGHT
  plate-list PRESS p1
  FLOOR WEIGHT
  YRANGE ...
```

**Note:** See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

- `cfe-spec` = { ZONE f1 QX f2 QZ f3 GROUP f4 STYP f5 (REGULAR) (TS f6) (PX f7) (PZ f8) }
  - **ZONE f1** Zone number specified in number such as 1, 2, 3 or 4
  - **QX f2** seismic behavior factor of the structure along X direction as a parameter according 3.2.4.
  - **QZ f3** seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.
  - **GROUP f4** Group of structure entered as A or B
  - **STYPE f5** Soil type entered as 1 or 2 or 3
  - **TS f6** site characteristic period
  - **PX f7** Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method
  - **PZ f8** Optional Period of structure (in sec) in Z direction (or Y if SET Z UP is used) to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

The optional parameter REGULAR is entered to consider the structure as a regular structure. By default, all structures are considered as irregular.
**Generation of CFE Seismic Load**

To provide a CFE Seismic load in any load case:

\[
\text{LOAD } i
\]

\[
\text{CFE LOAD } \{X \mid Y \mid Z\} (f)
\]

Where:

- \(\text{LOAD } i\) load case number
- \(\text{CFE LOAD } \{X \mid Y \mid Z\} (f)\) factor to multiply horizontal seismic load. Choose horizontal directions only

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

**Methodology**

Seismic zone coefficient and parameter values are supplied by the user through the `DEFINE CFE LOAD` command.

Program calculates the natural period of building \(T\) utilizing Rayleigh-Quotient method. If time period is provided in the input file, that is used in stead of calculated period.

The acceleration \(a\) is calculated according to the following:

\[
a = \begin{cases} 
  a_0 + \left( c - a_0 \right) \frac{T}{T_a} & \text{when } T < T_a \\
  c & \text{when } T_a \leq T \leq T_b \\
  c \left( \frac{T_b}{T} \right)^r & \text{when } T_b < T 
\end{cases}
\]

Where:

- \(c\) = Seismic coefficient is extracted from table 3.1
- \(a_0, T_a, T_b, r\) are obtained from table 3.1

The ductility reduction factor \(Q'\) is calculated according to section 3.2.5.

\[
Q' = \begin{cases} 
  Q & \text{when } T \geq T_a \\
  1 + \left( \frac{T}{T_a} \right) (Q - 1) & \text{when } T < T_a 
\end{cases}
\]

If not regular, then \(Q' = Q' \times 0.8\)

If the period \(T_s\) of the soil is known and the soil type II or III \(T_a\) and \(T_b\) will be modified according to section 3.3.2.

Lateral loads for each direction are calculated for:

When \(T \leq T_b\), Eq. 4.5. Section 3.4.4.2 is used:

\[
P_n = W_n \cdot h_n \cdot \sum_{n=1}^{N} \left( \frac{W_n h_n}{Q'} \right)^a
\]

When \(T > T_b\), Eq. 4.6/7/8. Section 3.4.4.2 is used:
\[ P_n = W_n(a/Q)(K_1 h_i + K_2 h_i^2) \]

Where:

\[ K_1 = \frac{q(1 - r(1 - q))\Sigma W_i}{2(W_i / h_i)} \]

\[ K_2 = \frac{1.5q(1 - q)\Sigma W_i}{2(W_i / h_i^2)} \]

The base shear are distributed proportionally to the height if \( T \leq T_b \) or with the quadratic equation mentioned if \( T > T_b \).

The distributed base shears are subsequently applied as lateral loads on the structure.

**Example**

```
UNIT KGS METER
DEFINE CFE LOAD
ZONE 2 QX .5 QZ 0.9 STYP 2 GROUP B TS 0.2
SELFWEIGHT
1 TO 36 41 TO 50 UNI 300
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
FLOOR WEIGHT
YRA 11.8 12.2 FLOAD 400 -
XRA -1 11 ZRA -1 21
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
CFE LOAD X 1.0
LOAD 2 ( SEISMIC LOAD IN -Z DIRECTION )
CFE LOAD Z -1.0
```

**Related Links**

- **G.16.2 Seismic Load Generator** (on page 2343)

**TR.31.2.18 NTC (Normas Técnicas Complementarias) Seismic Load**

The purpose of this command is to define and generate static equivalent seismic loads as per Code of the México Federal District (Reglamento de Construcciones del Distrito Federal de México) and Complementary Technical Standards for Seismic Design (y Normas Técnicas Complementarias (NTC) para Diseño por Sismo -Nov. 1987) (Chapters 8.1 8.2 8.6 and 8.8) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in **TR.5 Set Command Specification** (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.
**General Format**

```plaintext
DEFINE  NTC  LOAD

ntc-spec

SELFWEIGHT
JOINT  WEIGHT
joint-list
WEIGHT  w
MEMBER  WEIGHT
mem-list { UNI  v1  v2  v3  |  CON
ELEMENT  WEIGHT
plate-list  PRESS  p1
FLOOR  WEIGHT
YRANGE ...}
```

**Note:** See [TR.31.2.20 UBC 1994 or 1985 Load Definition](on page 2617) for complete weight specification and [TR.32.4 Area, One-way, and Floor Load Specifications](on page 2664) for FLOOR WEIGHT input description.

Where:

```plaintext
ntc-spec = { ZONE  f1  QX  f2  QZ  f3  GROUP  f4  (SHADOWED) (REGULAR) (REDUCE) (PX  f5) (PZ  f6) }
```

where:

- **ZONE f1**  Zone number specified in number such as 1, 2, 3 or 4
- **QX f2**  Seismic behavior factor of the structure along X direction as a parameter according 3.2.4.
- **QZ f3**  Seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.
- **GROUP f4**  Group of structure entered as A or B
- **PX f5**  Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method
- **PZ f6**  Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

**Generation of NTC Seismic Load**

To provide NTC Seismic load in any load case:

```plaintext
LOAD  i

NTC  LOAD  {X/Y/Z} (f)
```

where:
**Methodology**

The design base shear is computed in accordance with Sections 8.1 or 8.2 of the NTC as decided by the user.

**A.** Base Shear is given as

\[ \frac{V_o}{W_o} = c / Q \]

where

\[ c = \text{Seismic Coefficient, which is obtained by the program from the following table} \]

<table>
<thead>
<tr>
<th>Seismic Coefficient, c</th>
<th>Group A</th>
<th>Group B</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>0.24</td>
<td>0.16</td>
</tr>
<tr>
<td>II not shaded</td>
<td>0.48</td>
<td>0.32</td>
</tr>
<tr>
<td>III (and II where shaded)</td>
<td>0.60</td>
<td>0.40</td>
</tr>
</tbody>
</table>

\[ Q = \text{is entered by the user as a parameter} \]

**B.** Base shear is given as

\[ \frac{V_o}{W_o} = a / Q' \]

Where Reduction of Shear Forces are requested

Time Period \( T \) of the structure is calculated by the program based on using Rayleigh quotient technique.

you may override the period that the program calculates by specifying these in the input

\[ a = \left( 1 + \frac{3T}{T_a} \right)^c \text{when} T < T_a \]

\[ a = c \text{when} T_a \leq T \leq T_b \]

\[ a = q \cdot c \text{when} T_b < T \]

Where:

\[ q = \left( \frac{T_b}{T} \right)^r \]

\[ Q' = Q \text{when} T \geq T_a \]

\[ Q' = 1 + \left( \frac{T}{T_a} \right) \left( Q - 1 \right) \text{when} T < T_a \]

If not regular, then \( Q' = Q' \times 0.8 \)
Ta, Tb and r are taken from table 5-13 (Table 3.1 in the NTC).

Table 249: Values of Ta, Tb and r per NTC

<table>
<thead>
<tr>
<th>Zone</th>
<th>Ta</th>
<th>Tb</th>
<th>r</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>0.2</td>
<td>0.6</td>
<td>1/2</td>
</tr>
<tr>
<td>II not shaded</td>
<td>0.3</td>
<td>1.5</td>
<td>2/3</td>
</tr>
<tr>
<td>III (and II where shaded)</td>
<td>0.6</td>
<td>3.9</td>
<td>1.0</td>
</tr>
</tbody>
</table>

a shall not be less than c/4

Vo for each direction is calculated:

\[ V_o = \frac{W_o a}{Q} \text{ when } T \leq T_b \]
\[ V_o = \frac{\sum W_i a}{Q \left( K_1 h_i + K_2 h_i^2 \right)} \text{ when } T > T_b \]

Where:

\[ K_1 = \frac{q \left[ 1 - r \left( 1 - q \right) \right] \sum W_i}{\sum \left( W_i / h_i \right)} \]
\[ K_2 = \frac{1.5rq \left( 1 - q \right) \sum W_i}{\sum \left( W_i / h_i^2 \right)} \]

Wi and hi the weight and the height of the ith mass over the soil or embedment level.

The base shear are distributed proportionally to the height if \( T \leq T_b \) or with the quadratic equation mentioned if \( T > T_b \). The distributed base shears are subsequently applied as lateral loads on the structure.

Example

UNIT KGS METER
DEFINE NTC LOAD
ZONE 2 QX .5 QZ 0.9 GROUP B
SELFWEIGHT
ELEMENT WEIGHT
1577 TO 1619 PRESSURE 275
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
NTC LOAD X 1.0
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
NTC LOAD Z 1.0

Related Links
- **G.16.2 Seismic Load Generator** (on page 2343)

TR.31.2.19 Turkish Seismic Code

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per the specifications laid out in *Specification for Structures to be Built in Disaster Areas Part – III*
− Earthquake Disaster Prevention Amended on 2.7.1998, Official Gazette No. 23390 (English Translation). This is referred to as the Turkish Seismic Provisions.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

STAAD utilizes the following format to generate the lateral seismic loads.

```plaintext
DEFINE TURKISH LOAD
tur-spec

SELFWEIGHT
JOINT WEIGHT joint-list
WEIGHT w
MEMBER WEIGHT mem-list {
  UNI v1 v2 v3 | CON
  ELEMENT WEIGHT plate-list PRESS p1
  FLOOR WEIGHT
  YRANGE ...
}
```

Note: See TR.31.2.20 UBC 1994 or 1985 Load Definition (on page 2617) for complete weight specification and TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for FLOOR WEIGHT input description.

Where:

```plaintext
tur-spec = { A f1 TA f2 TB f3 I f4 RX f5 RZ f6 (CT f7) (PX f8) (PZ f9) }
```

- **A f1**: Effective Ground Acceleration Coefficient, Ao. Refer table 6.2
- **TA f2**: Spectrum Characteristic Periods, TA and TB. These are user input and found in Table 6.4
- **TB f3**: Spectrum Characteristic Periods, TA and TB. These are user input and found in Table 6.4
- **I f4**: the earthquake importance factor of the structure.
- **RX f5**: Structural Behavior Factors (R) along X and Z directions respectively. These are user input and please refer Table 6.5
- **RZ f6**: Structural Behavior Factors (R) along X and Z directions respectively. These are user input and please refer Table 6.5
- **CT f7**: Optional CT value to calculate time period. See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02.
- **PX f8**: Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.
- **PZ f9**: Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.
- **WEIGHT w**: joint weight associated with list
**UNI v1 v2 v3** Used when specifying a uniformly distributed load with a value of \( v1 \) starting at a distance of \( v2 \) from the start of the member and ending at a distance of \( v3 \) from the start of the member. If \( v2 \) and \( v3 \) are omitted, the load is assumed to cover the entire length of the member.

**CON v4 v5** Used when specifying a concentrated force with a value of \( v4 \) applied at a distance of \( v5 \) from the start of the member. If \( v5 \) is omitted, the load is assumed to act at the center of the member.

**PRESSURE p1** weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (See TR.32.4.3 Floor Load Specification (on page 2672) for details).

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2.1 Generation of Seismic Loads (on page 2773).

**Base Shear**

The minimum lateral seismic force or base shear, \( V_t \), is automatically calculated per equation 6.4 in section 6.7.1:

\[
V_t = \frac{WA(T_1)}{R_a(T_1)}
\]

where

\[ T_1 = \text{the fundamental time period of the structure} \]

Except that \( V_t \) shall not be less than:

\[ V_{t,\text{min}} = 0.10A_0IW \]

Seismic Load Reduction Factor, \( R_a(T_1) \), in above equation is determined based on the following equations 6.3a and 6.3b in the code:

\[
R_a(T_1) = \begin{cases} 
1.5 + \left( R - 1.5 \right) \frac{T_1}{T_a} & \text{when } 0 \leq T_1 \leq T_a \\
R & \text{when } T_1 > T_a 
\end{cases}
\]

The Structural Behavior Factor in either direction, \( RX \) and \( RZ \), are provided through user input (variables \( f5 \) and \( f5 \)) along the direction of calculation. Spectrum Characteristics Period, \( TA \) and \( TB \), are also provided by user through the parameters (variables \( f2 \) and \( f3 \)).

where

\[ R_a = \text{the Structural Behavior Factor in either direction, } RX \text{ and } RZ, \text{ are provided through user input (variables } f5 \text{ and } f5 \text{) along the direction of calculation} \]

\[ T_{a, b} = \text{the Spectrum Characteristics Period are provided by user through the parameters (variables } f2 \text{ and } f3 \text{)} \]

\[ T_1 = \text{the fundamental Lateral period in the direction under consideration and is determined as:} \]

\[ a. \text{ Calculated by the empirical formulae as described below provided } hn \text{ is in meter:} \]

\[ T_1 = C_T [hn] ^{\frac{1}{n}} \]
Where $C_T$ is assumed to be 0.075 for steel moment frames, 0.085 for concrete moment frames, or any user specified value.

b. The period is also calculated in accordance with the Rayleigh method but could be overridden by user specified time period (PX, PZ).

The time period calculated based on method (a) is used in further calculation unless it is greater than 1.0 sec and 1.3 times of this is greater than the same calculated based on method (b). In that case time period calculated based on method (b) is used.

\[
A(T_1) = \frac{T_1}{T_A}
\]

\[
A_0 I S(T_1)
\]

in above equation are Effective Ground Acceleration Coefficient and Building Importance Factor are provided by the user through the load definition parameter and could be found in table 6.2 and 6.3 respectively in the code

\[
S(T_1) = \begin{cases} 1 + 1.5 \frac{T_1}{T_A} & \text{when } 0 \leq T < T_A \\ 2.5 \text{when } T_A \leq T \leq T_B \\ 2.5 \left( \frac{T_B}{T_A} \right)^{0.8} & \text{when } T_B < T \\
\end{cases}
\]

$W$ is the weight of the building and shall be calculated internally using the following formula:

\[
W = \sum_{i=1}^{n} W_i
\]

$W_i$ is the portion of $W$ that is located at or assigned to level $i$.

Vertical Distribution

As per 4.1.8.11(6), the total lateral seismic force, $V_t$, shall be distributed such that a portion, $F_t$ shall be concentrated at the top of the building, where,

\[
\Delta F_N = 0.07 \cdot T_1 \cdot V_t
\]

but $\Delta F_N$ is not greater than 0.20$V_t$

and $\Delta F_N = 0$ when $H_N \leq 25$ m.

The remainder ($V - \Delta F_N$), shall be distributed along the height of the building, including the top level, in accordance with the following formula (per equation 6.9):

\[
F_i = (V_t - \Delta F_N) Wi / \sum wj Hj
\]

where

$F_i$ = the lateral force applied to level $i$

$\Delta F_N$ = the portion of $V_t$ to be concentrated at the top of the structure

$W_i, W_j$ = the portion of $W$ that is located at or assigned to level $i$ or $j$ respectively

$i$ = level $i$ is any level in the building, $i = 1$ for first level above the base.
Example

```
DEFINE TUR LOAD
A 0.40 TA 0.10 TB 0.30 I 1.4 RX 3.0 RZ 3.0
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
TUR LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

Related Links

- [G.16.2 Seismic Load Generator](#) (on page 2343)

## TR.31.2.20 UBC 1994 or 1985 Load Definition

This set of commands may be used to define the parameters for generation of UBC-type equivalent static lateral loads for seismic analysis. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

### General Format

```
DEFINE UBC (ACCIDENTAL) LOAD
{ ubc-1994-spec | ubc-1985-spec }
SELFWEIGHT (f11)
JOINT WEIGHT
joint-list WEIGHT w
MEMBER WEIGHT
mem-list { UNI v1 v2 v3 | CON v4 v5 }
ELEMENT WEIGHT
plate-list PRESS p1
FLOOR WEIGHT
floor-weight-spec
ONEWAY WEIGHT
oneway-weight-spec
REFERENCE LOAD { X | Y | Z }
Ri1 f11
```

Where:

```
ubc-1994-spec = { ZONE f1 I f2 RWX f3 RWZ f4 S f5 (CT f8) (PX f9) (PZ f10) }
ubc-1985-spec = { ZONE f1 K f6 I f2 (TS f7) }
```
**Technical Reference of STAAD Commands**

**TR.31 Definition of Load Systems**

---

*floor-weight-spec* See [TR.32.4.3 Floor Load Specification](#) for floor weight specification.

*oneway-weight-spec* See [TR.32.4.2 One-way Load Specification](#) for One-way load specification.

**Note:** The weight definitions must be in the order specified above. That is, selfweight, joint weight, member weight, element weight, and then floor weights. If one or more is not present, it can be skipped as long as the general order is preserved.

Where:

- **ZONE f1** The seismic zone coefficient (0.2, 0.3 etc.). Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.
- **lf2** The importance factor. Used for both 1985 and 1994 specifications.
- **RWX f3** (UBC 1994 spec only) Numerical co-efficient \( R_w \) for lateral load in Z-directions
- **RWZ f4** (UBC 1994 spec only) Numerical co-efficient \( R_w \) for lateral load in Z-directions
- **S f5** (UBC 1994 spec only) Site co-efficient for soil characteristics
- **K f6** (UBC 1985 spec only) Horizontal force factor
- **TS f7** (UBC 1985 spec only) Period of structure (in seconds) in the X-direction.
- **CT f8** (UBC 1994 spec only) Value of the term \( C_t \) which appears in the equation of the period of the structure per Method A. See [note f](#) (on page 2619) for details.
- **PX f9** (UBC 1994 spec only) Period of structure (in seconds) in the X-direction.
- **PZ f10** (UBC 1994 spec only) Period of structure (in seconds) in the Z-direction. Used for Y if the SET Z_UP command is used.
- **WEIGHT w** The joint weight associated with list
- **UNI v1 v2 v3** Used when specifying a uniformly distributed load with a value of \( v1 \) starting at a distance of \( v2 \) from the start of the member and ending at a distance of \( v3 \) from the start of the member. If \( v2 \) and \( v3 \) are omitted, the load is assumed to cover the entire length of the member.
- **CON v4 v5** Used when specifying a concentrated force with a value of \( v4 \) applied at a distance of \( v5 \) from the start of the member. If \( v5 \) is omitted, the load is assumed to act at the center of the member.
- **PRESS p1** The weight per unit area for the plates selected. Assumed to be uniform over the entire plate. Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.
- **Ri1** Identification number of a previously defined reference load case. See [TR.31.6 Defining Reference Load Types](#) (on page 2642)
- **f11** Magnification factor (required for reference loads).

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which you would use FLOOR LOADS (See [TR.32.4.3 Floor Load Specification](#) for details). Similarly, you can use the Oneway Weight command to specify a load path direction for the pressure on a region.

**Note:** For additional details on the application of a seismic load definition used to generate loads, refer to [TR.32.12.2.1 Generation of Seismic Loads](#) (on page 2773).
Notes

a. If the option ACCIDENTAL is used, the accidental torsion will be calculated per UBC specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, MEMBER WEIGHT, ELEMENT WEIGHT, FLOOR WEIGHT, and ONEWAY WEIGHT commands you have specified.

b. In ubc-spec for 1985 code, specification of TS is optional. If TS is specified, resonance co-efficient S is determined from the building period T and user provided TS using UBC equations. If TS is not specified, the default value of 0.5 is assumed.

c. By providing either PX or PZ or both, you may override the period calculated by STAAD for Method B of the UBC Code. The user defined value will then be used instead of the one recommended by UBC per equation 28.5 of UBC 94. If you do not define PX or PZ, the period for Method B will be calculated by the program per equation 28.5.

d. Some of the items in the output for the UBC analysis are explained below.

   CALC / USED PERIOD

   The CALC PERIOD is the period calculated using the Rayleigh method (Method B as per UBC code). For UBC in the x-direction, the USED PERIOD is PX. For the UBC in the z-direction, the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one substituted into the critical equation of the UBC code to calculate the value of C.

e. In the analysis for UBC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

f. If the value of Ct is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller than 2000 ksi, Ct is set to 0.02. If the average E is between 2000 & 10000 ksi, Ct is set to 0.03. If the average E is greater than 10,000 ksi, Ct is set to 0.035. If the building material cannot be determined, Ct is set to 0.035. Ct is in units of seconds/feet$^{3/4}$ or in units of seconds/meter$^{3/4}$. Ct < 0.42 if the units are in feet, and Ct > 0.42 if the units are in meter.

Philosophy

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate UBC equations (All symbols and notations are per UBC).

UBC 1994: Equation 1

\[ V = ZIC \cdot W \]

\[ Rw \]

UBC 1984: Equation 2

\[ V = ZIK \cdot C_S \cdot W \]

Base shear \( V \) may be calculated by STAAD using either the 1994 procedure (equation 1) or the 1985 procedure (equation 2). The user should use the appropriate "ubc-spec" (see General Format (on page 2617)) to instruct the program accordingly.
**Procedure Used by the Program**

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. You must specify seismic zone co-efficient and desired ubc-spec (1985 or 1994) following the DEFINE UBC LOAD command.
2. Program calculates the structure period T.
3. Program calculates C from appropriate UBC equation(s) utilizing T.
4. Program calculates V from appropriate equation(s). W is obtained from SELFWEIGHT, JOINT WEIGHT, MEMBER WEIGHT, ELEMENT WEIGHT, FLOOR WEIGHT, and ONEWAY WEIGHT commands specified following the DEFINE UBC LOAD command. The weight data must be in the order shown.

   **Note:** If both mass table data (SELFWEIGHT, JOINT WEIGHT, MEMBER WEIGHT, etc. options) and a REFERENCE LOAD are specified, these will be added algebraically for a combined mass.

5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per UBC procedures.

```
Example

DEFINE UBC LOAD
ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 2.5
49 TO 64 WEIGHT 1.25
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
FLOOR WEIGHT
_SLAB1 FLOAD 0.045
ONEWAY LOAD
_ROOF ONE 0.035 GY
```

**Related Links**

- [G.16.2 Seismic Load Generator](on page 2343)

**TR.31.2.21 UBC 1997 Load Definition**

This feature enables one to generate horizontal seismic loads per the UBC 97 specifications using a static equivalent approach. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command in TR.5 Set Command Specification (on page 2413)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.
General Format

```
DEFINE UBC (ACCIDENTAL) LOAD
ubc-1997-spec
seismic-weights
```

Where:

```
ubc-1997-spec = { ZONE f1 I f2 RWX f3 RWZ f4 STYPE f5 NA f6 NV f7 (CT f8) (PX f9) (PZ f10)}
```

Where:

- **ZONE f1**: Seismic zone coefficient. Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.
- **I f2**: Importance factor
- **RWX f3**: Numerical coefficient R for lateral load in X direction
- **RWZ f4**: Numerical coefficient R for lateral load in Z direction
- **STYPE f5**: Soil Profile type. Valid range of values are integers 1 through 5. These are related to the values shown in Table 16-J of the UBC 1997 code in the following manner:
  1. S\textsubscript{A}
  2. S\textsubscript{B}
  3. S\textsubscript{C}
  4. S\textsubscript{D}
  5. S\textsubscript{E}

  **Note**: The soil profile type S\textsubscript{F} is not supported.
- **NA f6**: Near source factor Na
- **NV f7**: Near source factor Nv
- **CT f8**: Optional CT value to calculate time period based on Method A (see Note 7 (on page 2622))
- **PX f9**: Optional Period of structure (in sec) in X-direction to be used in Method B
- **PZ f10**: Optional Period of structure (in sec) in Z-direction to be used in Method B

The seismic zone factor (ZONE) in conjunction with the soil profile type (STYPE), Near source factor (NA), and the Near source factor (NV), is used to determine the values of seismic coefficients Ca and Cv from Tables 16-Q and 16-R of the UBC 1997 code.

If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the UBC specifications. The value of the accidental torsion is based on the “center of mass” for each level. The “center of mass” is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

```
seismic-weights =
SELFWEIGHT
JOINT WEIGHT
joint-list
WEIGHT w
MEMBER WEIGHT
mem-list { UNI v1 v2 v3 | CON
ELEMENT WEIGHT
```
Methodology

The design base shear is computed in accordance with Section 1630.2.1 of the UBC 1997 code. The primary equation, namely, 30-4 of UBC 1997, as shown below, is checked.

\[ V = C_v I/(RT) \cdot W \]

In addition, the following equations are checked:

Equation 30-5 – The total design base shear shall not exceed

\[ V = 2.5 \cdot C_a I/R \cdot W \]

Equation 30-6 - The total design base shear shall not be less than

\[ V = 0.11 \cdot C_a I W \]

Equation 30-7 – In addition, for Seismic Zone 4, the total base shear shall also not be less than

\[ V = 0.8 \cdot ZN_v I/R \cdot W \]

For an explanation of the terms used in the above equations, please refer to the UBC 1997 code.

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE UBC LOAD command.

Procedure Used by the Program

Steps to calculate base shear are as follows:

1. Time Period of the structure is calculated based on clause 1630.2.2.1 (Method A) and 1630.2.2.2 (Method B).
2. You may override the period that the program calculates using Method B by specifying a value for PX or PZ (Items f9 and f10) depending on the direction of the UBC load. The specified value will be used in place of the one calculated using Method B.
3. The governing Time Period of the structure is then chosen between the above-mentioned two periods on the basis of the guidance provided in clause 1630.2.2.2.
4. From Table 16-Q and 16-R, Ca and Cv coefficients are calculated.
5. The Design Base Shear is calculated based on clause 1630.2.1 and distributed at each floor using the rules of clause 1630.5.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the UBC load (clause 1630.6). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. If the value of Ct is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller
than 2000 ksi, Ct is set to 0.02. If the average E is between 2000 & 10000 ksi, Ct is set to 0.03. If the average E is greater than 10,000 ksi, Ct is set to 0.035. If the building material cannot be determined, Ct is set to 0.035. Ct is in units of seconds/feet$^{\frac{1}{2}}$ or in units of seconds/meter$^{\frac{1}{2}}$. Ct < 0.42 if the units are in feet, and Ct > 0.42 if the units are in meter.

8. Due to the abstractness of the expression “Height above foundation,” in STAAD, height, $h$, is measured above supports. If supports are staggered all over the vertical elevations of the structure, it is not possible to calculate "h" if one doesn't have a clear elevation level from where to measure "h". Also, the code deals with distributing the forces only on regions above the foundation. If there are lumped weights below the foundation, it is not clear as to how one should determine the lateral forces for those regions.

### Example 1

```
DEFINE UBC LOAD
ZONE 0.38 I 1.0 STYP 2 RWX 5.6 RwZ 5.6 NA 1.3 NV 1.6 CT 0.037
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
MEMBER WEIGHT
12 17 24 UNI 25.7
FLOOR WEIGHT
YRA 9 11 FLOAD 200 XRA -1 21 ZR -1 41
ELEMENT WEIGHT
234 TO 432 PR 150
```

### Example 2

The following example shows the commands required to enable the program to generate the lateral loads. See TR.32.12 Generation of Loads (on page 2771) for this information.

```
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
UBC LOAD X 0.75
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
UBC LOAD Z 0.75
```

The UBC / IBC input can be provided in two or more lines using the continuation mark (hyphen) as shown in the following example:

```
DEFINE UBC ACCIDENTAL LOAD
ZONE 3.000 -
I 1.00 RWX 1.100 -
RwZ 1.200 STYP 5.000 NA 1.40 NV 1.50 CT -
1.300 PX 2.100 PZ 2.200
```

### Related Links

- [G.16.2 Seismic Load Generator](on page 2343)
- [TR.31.3 Definition of Wind Load](on page 2771)

### TR.31.3 Definition of Wind Load

This set of commands may be used to define some of the parameters for generation of wind loads on the structure. See TR.32.12 Generation of Loads (on page 2771) for the definition of wind direction and the possible surfaces to be loaded. G.16.3 Wind Load Generator (on page 2345) describes the two types of structures on which this load generation can be performed.
The general wind load generator can be used to generate lateral loads in the horizontal –X and Z (or Y if Z up)– directions only.

**Tip:** The graphical user interface can be used to automatically generate the appropriate intensity values via the ASCE-7: Wind Load dialog (on page 3039). See Persistence of Parameters used to Generate ASCE Wind Loads (on page 2629)

**General Format**

```
DEFINE WIND LOAD

TYPE j ( optional_comment )
{
  intensity-definition | code-parameters 
}

EXPOSURE e1 { JOINT joint-list | YRANGE f1 f2 | ZRANGE f1 f2 }
```

Repeat EXPOSURE command up to 98 times.

Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>TYPE j optional_comment</strong></td>
<td>wind load system type number (integer)</td>
</tr>
<tr>
<td></td>
<td>The optional comment is a text string comment or description used to help identify the wind load type.</td>
</tr>
<tr>
<td><strong>intensity-definition or code-parameters</strong></td>
<td>data is entered based on either custom or Russian code wind definitions. See Wind Intensity Definition (on page 2625) or See Russian Wind Loads (on page 2626)</td>
</tr>
<tr>
<td><strong>EXPOSURE e1, e2 ... em</strong></td>
<td>exposure factors. A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction. Limit: 99 factors. If the command EXPOSURE is not specified or if a joint is not listed in an Exposure, the exposure factor for those joints is chosen as 1.0.</td>
</tr>
<tr>
<td><strong>JOINT joint-list</strong></td>
<td>Joint list associated with Exposure Factor (joint numbers or TO or BY) or enter only a group name.</td>
</tr>
<tr>
<td>**YRANGE</td>
<td>ZRANGE f1 f2**</td>
</tr>
</tbody>
</table>
Wind Intensity Definition

For custom (including for ASCE 7) wind load definitions, the wind intensity at heights above ground are defined as follows:

\[ \text{intensity-definition} = \text{INTENSITY } p_1 \ p_2 \ p_3 \ \ldots \ pn \ \text{HEIGHT } h_1 \ h_2 \ h_3 \ \ldots \ hn \]

**Note:** These values are automatically generated for ASCE 7 wind loads when the ASCE-7: Wind Load dialog is used.

Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(P_1, P_2, \ldots P_n)</td>
<td>wind intensities (pressures) in force/area. Up to 100 different intensities can be defined in the input file per type.</td>
</tr>
<tr>
<td>(h_1, h_2, h_3, \ldots h_n)</td>
<td>corresponding heights in global vertical direction, measured in terms of actual Y (or Z for Z UP) coordinates up to which the corresponding intensities occur.</td>
</tr>
</tbody>
</table>

All intensities and heights are in current unit system. The heights specified are in terms of actual Y coordinate ( or Z coordinates for Z UP) and not measured relative to the base of the structure. The first value of intensity \((p_1)\) will be applied to any part of the structure for which the Y coordinate ( or Z coordinate for Z UP) is equal to or less than \(h_1\). The second intensity \((p_2)\) will be applied to any part of the structure that has vertical coordinates between the first two heights \((h_1 \text{ and } h_2)\) and so on. Any part of the structure that has vertical coordinates greater than \(h_n\) will be loaded with intensity \(p_n\).

Only exposed surfaces bounded by members (not by plates or solids) will be used. The joint influence areas are computed based on surface member selection data entered in TR.32.12.3 Generation of Wind Loads (on page 2779) and based on the wind direction for a load case. Only joints actually exposed to the wind and connected to members will be loaded. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

Exposure factor \((e)\) is the fraction of the influence area associated with the joint(s) on which the load may act if it is also exposed to the wind load. Total load on a particular joint is calculated as follows.

\[ \text{Joint load} = (\text{Exposure Factor}) \times (\text{Influence Area}) \times (\text{Wind Intensity}) \]

The exposure factor may be specified by a joint-list or by giving a vertical range within which all joints will have the same exposure. If an exposure factor is not entered or not specified for a joint, then it defaults to 1.0 for those joints; in which case the entire influence area associated with the joint(s) will be considered.

For load generation on a closed type structure defined as a PLANE FRAME, influence area for each joint is calculated considering unit width perpendicular to the plane of the structure. You can accommodate the actual width by incorporating it in the Exposure Factor as follows.

\[ \text{Exposure Factor (User Specified)} = (\text{Fraction of influence area}) \times (\text{influence width for joint}) \]

**Notes:**

a. All intensities, heights and ranges must be provided in the current unit system.
b. If necessary, the INTENSITY and EXPOSURE command lines can be continued on to additional lines by ending all but last line with a space and hyphen (-). Use up to 11 lines for a command.

Example

UNIT FEET
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.15 HEIGHT 12 24
EXPOSURE 0.90 YRANGE 11 13
EXPOSURE 0.85 JOINT 17 20 22
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1

Note: For additional examples, see TR.32.12 Generation of Loads (on page 2771) and EX. US-15 Wind and Floor Load Generation on a Space Frame (on page 4480).

The Intensity line can be continued in up to 12 lines.

So the following

INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012 0.012 0.012 HEIG 15 20 25 30 40 50 60 70 80 90 100 120

could be split as

INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012 0.012 0.012 – HEIG 15 20 25 30 40 50 60 70 80 90 100 120

or

INT 0.008 0.009 0.009 0.009 0.01 0.01 -0.01 0.011 0.011 0.012 0.012 0.012 HEIG 15 20 25 -30 40 50 60 70 80 90 100 120

e tc.

Russian Wind Loads

This specifies the definition of a wind load to the Russian wind code which will need to be referenced in a wind load command included in a primary load case. For wind loads per Russian codes SNiP 85, SP 20 2011, or SP20 2016, the code parameters are defined as follows:

\[
\text{code-parameters} = \{ \text{SNIP 1985 } | \text{SNIP 2011 } \} \text{PRESSURE } f3 \text{ TERRAIN } \{ \text{A } | \text{B } | \text{C } \} \text{CLASSIFICATION } \{ \text{1 } | \text{2 } | \text{3 } \}
\]

or

\[
\text{code-parameters} = \{ \text{SP20 } \} \text{PRESSURE } f3 \text{ TERRAIN } \{ \text{A } | \text{B } | \text{C } \} \text{CLASSIFICATION } \{ \text{1 } | \text{2 } | \text{3 } \} \text{REGION } f4 \text{ LOG } f5
\]

Where:

SNIP 1985 — design according to previous design Code SNiP 2.01.07-85
SNIP 2011 — design according to renewed design Code SP 20.13330.2011
SP20 2016 — design according to renewed design Code SP 20.13330.2016
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRESSURE ( f_3 )</td>
<td>the characteristic value of wind pressure, always positive</td>
</tr>
<tr>
<td>TERRAIN</td>
<td>terrain roughness category:</td>
</tr>
<tr>
<td></td>
<td>A. Coastal Zone.</td>
</tr>
<tr>
<td></td>
<td>B. Urban Zone.</td>
</tr>
<tr>
<td></td>
<td>C. Large City.</td>
</tr>
<tr>
<td>CLASSIFICATION</td>
<td>the type of structure:</td>
</tr>
<tr>
<td></td>
<td>1. Prismatic building structures,</td>
</tr>
<tr>
<td></td>
<td>2. General-type concrete structures,</td>
</tr>
<tr>
<td></td>
<td>3. General-type framed steel structures.</td>
</tr>
<tr>
<td>REGION ( f_4 )</td>
<td>Wind region as per clause 11.5 of SNiP 2.01.07-85* 2016. This is used to determine the wind pressure in determine the dynamic wind component.</td>
</tr>
<tr>
<td></td>
<td>0 = region 1a</td>
</tr>
<tr>
<td></td>
<td>1 = region 1</td>
</tr>
<tr>
<td></td>
<td>2 = region 2</td>
</tr>
<tr>
<td></td>
<td>3 = region 3</td>
</tr>
<tr>
<td></td>
<td>4 = region 4</td>
</tr>
<tr>
<td></td>
<td>5 = region 5</td>
</tr>
<tr>
<td></td>
<td>6 = region 6</td>
</tr>
<tr>
<td></td>
<td>7 = region 7</td>
</tr>
<tr>
<td>LOG ( f_5 )</td>
<td>logarithmic decrement of oscillations, Delta (see table 11.5 - Section 11.1.8 for the definition). Typically values are:</td>
</tr>
<tr>
<td></td>
<td>0.15 for steel towers, masts, lined chimneys, column means including the ones on the reinforced concrete pedestals</td>
</tr>
<tr>
<td></td>
<td>0.3 for reinforced concrete and stone structures as well as for buildings with steel framework if there are walling structures</td>
</tr>
</tbody>
</table>

**Notes:**

a. A structured modeled as a set of vertical members—such as is typically used to define a cylindrical stack or chimney structure, and thus does not define a closed panel such as defined by a frame formed from a number of columns and beams—is considered to be a “Stick Structure.” Refer to TR.32.12.3 Generation of Wind Loads (on page 2779) for additional details.

b. The loading has two components: static and dynamic. The dynamic effect is determined by the number of modes included in the dynamic load case and changing the number of modes considered using the CUT.
OFF MODE SHAPE command may result in a change in the resulting wind force. Refer to TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2539) for details.

As the analysis will require extraction of eigen solutions to determine the dynamic effects of the wind loading, the number of modes to be used will also affect the results. Thus setting the command CUT OFF FREQUENCY or CUT OFF MODE should be considered and specified as required prior to the definition of the load cases.

If the cut-off command is omitted, six mode shapes are computed by default.

c. If the PRINT STATICS CHECK option is included in the analysis command, then the output file will include a section on the SNiP wind load. This output will indicate both the static and dynamic contributions to the wind load as well as the total applied at each node.

d. This command cannot be used with models that have been defined with the SET Z UP command.

The Russian wind load generation commands (on page 2781) must also be used in conjunction with the SNiP wind load definition.

STAAD.Pro can generate both static and dynamic wind loads per the SP 20.13330.2016 code.

Example

An example with various static wind loads from different directions that use a SNiP 1985 definition:

```
DEFINE WIND LOAD
TYPE 1
SNIP 1985 PRESSURE 0.38 TERRAIN A CLASSIFICATION 1
EXP 0.5 JOINT 1 3 5 7 9 11

* LOAD 1 LOADTYPE WIND TITLE Wind load in the +ve X direction
WIND LOAD X 1 CONFIG 0 NU 1 TYPE 1
* Mass model required in first wind load case
JOINT LOAD
3 TO 6 FZ 62.223
9 TO 12 FZ 62.223

* LOAD 2 LOADTYPE WIND TITLE Wind load in the -ve X direction
WIND LOAD X -1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case

* LOAD 3 LOADTYPE WIND TITLE Wind load in the +ve Z direction
WIND LOAD Z 1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case

* LOAD 4 LOADTYPE WIND TITLE Wind load in the -ve Z direction
WIND LOAD Z -1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case

* LOAD 10 LOADTYPE DEAD TITLE Selfweight load case
SELF Y -1 ALL

* LOAD COMBINATION 100 Wind plus selfweight
1.0 1.0
```

The following example uses SP 20.13330.2016 with both static and dynamic load cases:

```
DEFINE WIND LOAD
TYPE 1
```
Persistence of Parameters used to Generate ASCE Wind Loads

In the Analytical Modeling workflow, in the Add New: Wind Definition dialog on page 3038, you can click the Calculate as per ASCE-7 button to generate the pressure versus height table per the ASCE 7 wind load specifications per the 1995, 2002, or 2010 editions. The parameters which go into the derivation of this table are not retained by the graphical environment but rather added into the STAAD input file so they may be edited as needed. These values are not read by the STAAD engine directly and, therefore, are not directly processed as a load but are rather used to generate the wind intensity values which are used by the engine. An example of it is shown below.

Example

DEFINE WIND LOAD
TYPE 1
This entire section of the input file must not be edited.<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2002:PARAMS 85.000 MPH 0 1 0 0 0.000 FT 0.000 FT 0.000 FT 1 -
1 40.000 FT 30.000 FT 25.000 FT 2.000 0.010 0 -
0 0 0 0.761 1.000 0.870 0.850 0 -
0 0 0 0.866 0.800 0.550
!> END GENERATED DATA BLOCK
INT 0.0111667 0.0111667 0.0113576 0.0115336 0.0116972 0.0118503 0.0119944 -
0.0121307 0.0122601 0.0123834 0.0125012 0.0126141 0.0127226 0.0128270.0129277 -
TR.31.4 Definition of Time History Load

This set of commands may be used to define parameters for Time History loading on the structure. The time history data may be specified using either explicit definition, function specification, a spectrum specification, or time history data provided in an external file.

General Format

```
DEFINE TIME HISTORY (DT x) (MISSING MASS)
```

```
TYPE i { ACCELERATION | FORCE | MOMENT } (SCALE f7) (SAVE)
```

Repeat TYPE and Amplitude vs Time sets until all are entered, then:

```
ARRIVAL TIME
```

```
a_1 a_2 a_3 ... a_n
```

```
{ DAMPING d | CDAMP | MDAMP }
```

Entering the `MISSING MASS` parameter will include the missing mass procedure in the time history analysis.

The time history data can be explicitly defined using pairs of time and values of acceleration, force, or moment, where:

- **ACCELERATION** indicates that the time varying load type is a ground motion.
- **FORCE** indicates that it is a forcing function.
- **MOMENT** indicates that it is a moment forcing function.

Table 251: Parameters for explicitly defined time history load

<table>
<thead>
<tr>
<th>Variable or Command</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DT x</td>
<td>-</td>
<td>Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.</td>
</tr>
<tr>
<td>TYPE i</td>
<td>-</td>
<td>Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.</td>
</tr>
<tr>
<td>Variable or Command</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SCALE $f_7$</td>
<td>1.0</td>
<td>The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).</td>
</tr>
<tr>
<td>SAVE</td>
<td>-</td>
<td>The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: TYPE 1 FORCE SAVE</td>
</tr>
<tr>
<td>$t_1 \ p_1 \ t_2 \ p_2$</td>
<td>-</td>
<td>Values of time (in sec.) and corresponding force (current force unit) or acceleration (current length unit/sec$^2$) depending on whether the time varying load is a forcing function or a ground motion. <strong>Note:</strong> If the data is specified through the input file, up to 499 pairs can be provided for each type in the ascending value of time. More than one line may be used if necessary. However, if the data is provided through an external file, an unlimited number of time-force pairs may be specified.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If the first point is not at zero time, then the forces before the first time (but after the arrival time) will be determined by extrapolation using the first two points entered. If the first point has a nonzero force, there will be a sudden application of that force over a single integration step (DT) at that time. Zero force will be assumed for all times after the last data point.</td>
</tr>
</tbody>
</table>
The function-spec option can be used to specify harmonic loads. Both sine and cosine harmonic functions may be specified. The program will automatically calculate the harmonic load time history based on the following specifications.

For the function and amplitude option :

```
function-spec =
{ SINE | COSINE }
AMPLITUDE f0 { FREQUENCY | RPM } f2 (PHASE f3) CYCLES f4 { SUBDIV f5 | STEP f6 }
```

**Note:** Please be aware that if a Cosine function or Sine with nonzero phase angle is entered, the force at the arrival time will be nonzero; there will be a sudden application of force over a single integration step (DT) at that time.

Where:
### Table 252: Parameters for use with function time history load

<table>
<thead>
<tr>
<th>Variable or Command</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DT (x)</td>
<td>-</td>
<td>Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.</td>
</tr>
<tr>
<td>TYPE (i)</td>
<td>-</td>
<td>Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.</td>
</tr>
<tr>
<td>SCALE (f7)</td>
<td>1.0</td>
<td>The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).</td>
</tr>
<tr>
<td>SAVE</td>
<td>-</td>
<td>The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: TYPE 1 FORCE SAVE</td>
</tr>
<tr>
<td>(f_0)</td>
<td>-</td>
<td>Max. Amplitude of the forcing function in current units.</td>
</tr>
<tr>
<td>(f_2)</td>
<td>-</td>
<td>If FREQUENCY, then cyclic frequency (cycles / sec.) If RPM, then revolutions per minute.</td>
</tr>
<tr>
<td>(f_3)</td>
<td>0</td>
<td>Phase Angle in degrees.</td>
</tr>
<tr>
<td>(f_4)</td>
<td>-</td>
<td>No. of cycles of loading.</td>
</tr>
<tr>
<td>(f_5)</td>
<td>3</td>
<td>Used to subdivide a (\frac{1}{4}) cycle into this many integer time steps. <strong>Note:</strong> Only used to digitize the forcing function. It is not the DT used to integrate for the responses. More subdivisions will make the digitized force curve more closely match a sine wave. The default is usually adequate.</td>
</tr>
<tr>
<td>Variable or Command</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| \( f_6 \)           | \( (p/12) \)  | Time step of loading. Default is equal to one twelfth of the period corresponding to the frequency of the harmonic loading. (It is best to use the default)  

**Note:** Only used to digitize the forcing function. It is not the DT used to integrate for the responses. More subdivisions or smaller step size will make the digitized force curve more closely match a sine wave. |
| \( a_1 \ a_2 \ a_3 \ldots \ a_n \) | - | Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here.  

The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767)). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767) for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified. |
| DAMPING \( d \) | 0.05 | The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.  

If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)  

If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510) |

The `spectrum-spec` option can be used to specify a synthetic ground motion acceleration time history based statistically on a user supplied acceleration spectrum.  

The program will automatically calculate the acceleration time history based on the following specifications. Enter \( f_{12} \), \( f_{13} \), and \( f_{14} \) to indicate the rise, steady, & decay times, respectively.
For the spectrum option:

\[ \text{spectrum-spec} = \]

\[
\text{SPECTRUM TMAX f_9 DTI f_{18} DAMP f_{11} T1 f_{12} T2 f_{13} T3 f_{14} SEED f_{15}}
\]

\[
\text{OPTIONS NF f_{16} NITR f_{17} ( THPRINT f_{18} ) ( SPRINT f_{19} ) ( FREQ )}
\]

Starting on the next line, enter Spectra in the following input form:

\[ P_1, V_1; P_2, V_2; \ldots; P_n, V_n \]

Where:

**Table 253: Parameters used with the spectrum time history load**

<table>
<thead>
<tr>
<th>Variable or Command</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DT x</td>
<td>0.0013888</td>
<td>Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.</td>
</tr>
<tr>
<td>TYPE i</td>
<td>1.0</td>
<td>Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.</td>
</tr>
<tr>
<td>SCALE f7</td>
<td>1.0</td>
<td>The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).</td>
</tr>
<tr>
<td>SAVE</td>
<td>-</td>
<td>The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: TYPE 1 FORCE SAVE</td>
</tr>
<tr>
<td>f_9</td>
<td>20 seconds</td>
<td>The Max. time (in seconds) in the generated time history.</td>
</tr>
<tr>
<td>f_{10}</td>
<td>0.2</td>
<td>Delta time step (in seconds) in the generated time history.</td>
</tr>
<tr>
<td>f_{11}</td>
<td>0.05</td>
<td>Damping ratio (5% is entered as 0.05) associated with the input spectrum.</td>
</tr>
<tr>
<td>f_{12}</td>
<td>4 seconds</td>
<td>Ending time of the acceleration rise time.</td>
</tr>
<tr>
<td>f_{13}</td>
<td>9 seconds</td>
<td>Ending time of the steady acceleration.</td>
</tr>
<tr>
<td>f_{14}</td>
<td>14 seconds</td>
<td>Ending time of the acceleration decay.</td>
</tr>
<tr>
<td>Variable or Command</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>$f_{15}$</td>
<td>15</td>
<td>Random Seed. Enter a positive integer (1 to 2147483647) to be used as a unique random number generation “seed.” A unique time history will be produced for each seed value. Change this value when you want to produce a “different (from the time history generated with the prior seed value)” but statistically equivalent time history. Omit this entry to get the default value (normal option).</td>
</tr>
<tr>
<td>$f_{16}$</td>
<td></td>
<td>The input shock spectrum will be re-digitized at $NF$ equally spaced frequencies by interpolation. Default is the greater of 35 or the number of points in the input spectrum.</td>
</tr>
<tr>
<td>$f_{17}$</td>
<td>10</td>
<td>The number of iterations which will be used to perfect the computed time history.</td>
</tr>
</tbody>
</table>
| $f_{18}$            | 1             | Print the time history that is generated. Omit the THPRINT parameter to avoid printing.  
1 = print beginning 54 values and last 54 values  
2 = Print entire curve.  
>10 = print beginning $f_{18}$ values and last $f_{18}$ values |
| $f_{19}$            | 1             | Print the spectrum generated from the time history that is generated. Omit the SPRINT parameter to avoid printing. |
| FREQ                |               | If entered, then frequency-spectra pairs are entered rather than period-spectra pairs. |
| $P_1, V_1$          |               | Data is part of input, immediately following SPECTRUM command. Period (or frequency if FREQ option entered above)  
Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Enter the period in seconds (or frequency in Hz.) and the corresponding Value is in acceleration (current length unit/sec$^2$) units. Continue the curve data onto as many lines as needed (up to 999 spectrum pairs). Spectrum pairs must be in ascending or descending order of period (or frequency).  
**Note:** If data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.807, 386.1, etc.). Also note, do not end these lines with a hyphen. Commas and semi-colons are optional. |
| $P_2, V_2$          |               |  
...  
$P_n, V_n$          |               |  

### Variable or Command | Default Value | Description
--- | --- | ---
\(a_1\), \(a_2\), \(a_3\), ..., \(a_n\) | - | Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here.

The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767)). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767) for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.

**DAMPING** \(d\) | 0.05 | The damping ratio. Specify a value of exactly 0.00000011 to ignore damping.

If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)

If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510)

---

**Note:** Please be aware that if a Cosine function or Sine with nonzero phase angle is entered, the force at the arrival time will be nonzero; there will be a sudden application of force over a single integration step (DT) at that time.

The time history data can also be defined in an external file, where:

**Table 254: Parameters for a time history load defined in an external file**

<table>
<thead>
<tr>
<th>Variable or Command</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DT (x)</td>
<td>-</td>
<td>Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.000001 will be reset to the default DT value of 0.0013888 seconds.</td>
</tr>
<tr>
<td>TYPE (i)</td>
<td>-</td>
<td>Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.</td>
</tr>
<tr>
<td>Variable or Command</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SCALE f7</td>
<td>1.0</td>
<td>The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).</td>
</tr>
<tr>
<td>SAVE</td>
<td>-</td>
<td>The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: TYPE 1 FORCE SAVE</td>
</tr>
<tr>
<td>READ filename</td>
<td></td>
<td>Filename for an external file containing time varying load history data.</td>
</tr>
<tr>
<td>f8</td>
<td></td>
<td>The optional delta time spacing used for the external file data</td>
</tr>
<tr>
<td>a1 a2 a3 ... an</td>
<td>-</td>
<td>Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here. The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767)). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to TR.32.10.2 Time Varying Load for Response History Analysis (on page 2767) for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.</td>
</tr>
</tbody>
</table>
### Variable or Command | Default Value | Description
--- | --- | ---
DAMPING $d$ | 0.05 | The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.

If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to [TR.26.2 Specifying Constants for Members and Elements](on page 2503)

If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to [TR.26.4 Modal Damping Information](on page 2510)

#### Examples

Using Force and Acceleration options:

```plaintext
UNIT ...
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
TYPE 2 ACCELERATION SCALE 9.80665
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
```

Using the Spectrum option:

```plaintext
UNIT ...
DEFINE TIME HISTORY
TYPE 1 ACCELERATION SCALE 9.80665
SPECTRUM TMAX 19 DTI 0.01 DAMP 0.03
OPTIONS NF 40
0.03 1.00 ; 0.05 1.35
0.1 1.95 ; 0.2 2.80
0.5 2.80 ; 1.0 1.60
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
```

Using the Harmonic Loading Generator:

```plaintext
UNIT ...
DEFINE TIME HISTORY
TYPE 1 FORCE
*Following lines for Harmonic Loading Generator
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100 STEP 0.02
ARRIVAL TIME
0.0
DAMPING 0.075
```
To define more than one sinusoidal load, the input specification is as follows:

```
DEFINE TIME HISTORY
TYPE 1 FORCE
FUNCTION SINE
AMPLITUDE 1.925 RPM 10794.0 CYCLES 1000
TYPE 2 FORCE
FUNCTION SINE
AMPLITUDE 1.511 RPM 9794.0 CYCLES 1000
TYPE 3 FORCE
FUNCTION SINE
AMPLITUDE 1.488 RPM 1785.0 CYCLES 1000
ARRIVAL TIME
0.0 0.0013897 0.0084034
DAMPING 0.04
```

The data in the external file must be provided as one or more time-force pairs per line as shown in the following example.

Data in Input file

```
UNIT ...
DEFINE TIME HISTORY
TYPE 1 FORCE
READ THFILE
ARRIVAL TIME
0.0
DAMPING 0.075
```

Data in the External file THFILE:

```
0.0 1.0 1.0 1.2
2.0 1.8
3.0 2.2
4.0 2.6
```

Notes

a. By default the response (displacements, forces etc.) will contain the contribution of only those modes whose frequency is less than or equal to 108 cps. Use the CUT OFF FREQUENCY command to change this limit. Contributions of modes with frequency greater than the Cut Off Frequency are not considered.

b. Results are the individual maximums over the time period. Thus, derived quantities such as section forces and stresses, plate surface stresses and principal stresses should not be used.

c. Results from harmonic input are the maximum over the time period including the start-up transient period. These results are not the steady-state results.

d. By default, the results do not include the time period after the time loads end. Use the CUT OFF TIME command to lengthen (or shorten) the time period. If an intense short-term loading is used, the loading should be continued until after the expected peak response is reached.

e. The READ filename command is to be provided only if the history of the time varying load is to be read from an external file. filename is the file name and may be up to 72 characters long. If the data on the file consists only of amplitudes, then enter f8 as the delta time spacing.

Related Links

- G.17.3.5 Response Time History (on page 2373)
- M. To define a time history type from tabular data (on page 855)
- M. To define a time history type from a function (on page 856)
TR.31.5 Definition of Snow Load

This set of commands may be used to define some of the parameters for generation of snow loads on the structure.

Refer to TR.32.13 Generation of Snow Loads (on page 2785) or the definition of additional parameters and the surfaces to be loaded.

**General Format**

```
DEFINE SNOW LOAD
TYPE f1 PG f2 CE f3 CT f4 IM f5
```

Where:

**TYPE f1**  Type No: (limit of 100). The “Type No.” is an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model. Include as many types as needed.

**PG f2**  Ground Snow Load (Default = 0.0). The pressure or, weight per unit area, to be used for the calculation of the design snow load. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of SEI/ASCE 7-02.

**CE f3**  Exposure Factor (Default = 1.0). Exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.

**CT f4**  Thermal Factor (Default = 0.0). Thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.

**IM f5**  Importance Factor (Default = 1.0). Importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

**Example**

```
START GROUP DEFINITION
FLOOR
  ROOFSNOW 102 TO 153 159 160 TO 170 179 195 TO 197
END GROUP DEFINITION
UNIT FEET POUND
```
TR.31.6 Defining Reference Load Types

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. Thus Reference Loads may be defined for this purpose. This is similar to a REPEAT LOAD command (See TR.32.11 Repeat Load Specification (on page 2770)), but has the added benefit of not being solved in its own right.

This converts a real load case to something similar to a load case definition. A reference load case is solved only when it is later called in a load case. The benefit is that it enables you to define as many load cases as you wish, but instruct the program to actually solve only a limited number of "real" load cases, thus limiting the amount of results to be examined.

Note: This feature requires STAAD.Pro 2007 Build 01 or higher.

See TR.33 Reference Load Cases - Application (on page 2789) for a description of the procedure for specifying the reference load information in active load cases.

General Format

```
DEFINE REFERENCE LOADS
LOAD R(i) LOADTYPE (type) TITLE load_title
   (Load items)
...
END DEFINE REFERENCE LOADS
```

Example

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead TITLE REF DEAD
   SELFWEIGHT Y -1
   JOINT LOAD
   4071 4083 4245 4257 FY -4.04
   4090 FY -0.64
   ELEMENT LOAD
   378 TO 379 406 TO 410 422 TO 426 PR GY -1.44
   MEMBER LOAD
   5006 TO 5229 UNI GY -0.64
   PMEMBER LOAD
   1 TRAP GY -0.347 -0.254 35.5 42
LOAD R2 LOADTYPE Live TITLE REF LIVE
   JOINT LOAD
   4209 FY -6.63
   4071 4083 4245 4257 FY -1.71
```
Mass Modeling Using Reference Loads

A reference load case of type MASS can be created which can then be used to define the structure mass used for all dynamic analyses (i.e., seismic, response spectrum, time history, etc.). Some analysis methods require you to create separate weight tables in the form of SELFWEIGHT, MEMBER WEIGHT, JOINT WEIGHT, etc. for each analysis, thus resulting in repetition of the same information. Using a LOADTYPE MASS reduces the repetitive data entry and the need for manually creating a weight table.

A mass model using this method is defined once and then used for all dynamic analyses.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) or higher.

If the LOADTYPE MASS is missing and no mass is defined in the corresponding seismic or dynamic analysis load cases, the program will report error as mass is missing. If a mass table is provided in a seismic load definition, response spectrum definition, or time history loading definition, then that mass table will be used for analysis under those loads only instead of a mass model generated by a REF LOAD TYPE MASS. However, if no masses are defined within the individual seismic, response spectrum, or time history load definitions, then the program uses the mass reference load case when analyzing that seismic or dynamic load case.

**Related Links**
- M. To add a mass model reference load (on page 876)
- M. To add mass loads to the mass model reference load (on page 877)
- M. To add weight by a reference load to a seismic load definition (on page 841)
- M. To create a reference load (on page 873)
- G.17.3.2 Mass Modeling (on page 2365)
- G.17.3.2 Mass Modeling (on page 2365)
- G.17.3.2 Mass Modeling (on page 2365)

**TR.31.7 Definition of Direct Analysis Members**

This set of commands may be used to define the members whose flexural stiffness or axial stiffness is considered to contribute to the lateral stability of the structure. These parameters are then used by the program to generate a set of lateral loads as defined in Appendix 7 of ANSI/AISC 360-05 when a PERFORM DIRECT ANALYSIS command is used.
General Format

**DEFINE DIRECT**

**FLEX** \((f1)\) **list-spec**

**AXIAL** **list-spec**

**FYLD** \((f2)\) **list-spec**

Where:

\[\text{list-spec} = * \{ \text{XR} \ f4 \ f5 \ | \ \text{YR} \ f4 \ f5 \ | \ \text{ZR} \ f4 \ f5 \ | \ \text{MEMBERS} \ \text{mem-list} \ | \ \text{LIST} \ \text{memb-list} \ | \ \text{ALL} \} \]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>FLEX</strong> (f1)</td>
<td>1.0</td>
<td>The initial (\tau_b) value (See Appendix 7 ANSI/AISC 360-05). Members listed with FLEX will have their EI factored by 0.80 times (\tau_b) while performing the global solution.</td>
</tr>
<tr>
<td><strong>AXIAL</strong></td>
<td>-</td>
<td>Members listed with AXIAL will have their EA factored by 0.80 while performing the global solution. Although reduced stiffness is considered for analysis, the program considers the full value of (E) for both flexural and axial design.</td>
</tr>
<tr>
<td><strong>FYLD</strong> (f2)</td>
<td>36.0 ksi</td>
<td>Yield Strength in current units.</td>
</tr>
<tr>
<td>**XR</td>
<td>YR</td>
<td>ZR** (f4, f5)</td>
</tr>
</tbody>
</table>

For specifying NOTIONAL LOADs, please see TR.32.14 Notional Loads (on page 2785). The notional loads and the factor used is specified entirely in the loading data.

**Notes**

\(\tau_b\) is the value entered in the FLEX command. \(\tau_b\) defaults to 1.0 if not entered.

**Related Links**

- [G.17.2.1.4 AISC 360 Direct Analysis](on page 2352)
- [M. To define direct analysis parameters](on page 852)
- [G.17.2.1.4 AISC 360 Direct Analysis](on page 2352)

**TR.31.8 Mass Modeling**

Each load case consisting of a response spectrum requires a mass definition. The following methods may be used to define masses for use with seismic loads STAAD.Pro.

**TR.31.8.1 Explicitly Defined Weights**

This method uses different load items within the load case to define the weights for use with that specific response spectrum. Combinations of selfweight, joint weights, member weights, element weights, and floor weights can defined prior to the response spectrum parameters.
Static Seismic Weight Definition Table

```
seismic-weights = 
SELFWEIGHT (f1) 
JOINT WEIGHT 
joint-list WEIGHT w 
MEMBER WEIGHT 
member-list \{ UNI v1 v2 v3 | CON v4 v5 \} 
ELEMENT WEIGHT 
plate-list PRESS p1 
FLOOR WEIGHT
floor-weight-spec 
ONETAY WEIGHT 
oneway-weight-spec 
REFERENCE LOAD \{ X | Y | Z \} 
Rl1 f2
```

Response Spectrum Weight Table

```
SELFWEIGHT \{ X | Y | Z \} f1 
JOINT LOADS 
joint-loads 
MEMBER LOADS 
member-loads
```

Example

TR.31.8.2 Reference Load Mass Tables

This method uses loads defined in a reference load case as the weights for the response spectrum. You may also define in which global direction the loading is considered for base shear calculation. By default, the program will consider Global Y if no load direction is specified. If no applied loads are found in the Y direction, then it consider the maximum of the total loads in either X or Z directions.

General Format

Multiple reference load cases can be used in a single response spectrum, each with a separate load factor applied. This allows you to factor the reference loads for use with weights. For example, if a code-prescribed seismic load requires that 100% of the dead load mass but only 50% of the live load mass be used for seismic loads, then you can use factors of 1.0 for the reference dead load case and 0.5 for the reference live load case.

This has the advantage of allowing you to define a single mass model for use with both static seismic loading as well as response spectra. In order to accomplish this, simply define the response spectrum load case as a reference load and then utilize this reference load in both the static seismic and response spectrum primary load case.

Tip: You may also use the same reference load cases for modal analysis and calculating Rayleigh’s frequency.
TR.31.8.3 Mass Model Using Reference Load

Mass type reference loads can be used to create mass model which will be used for all types of analysis, including seismic, response spectrum, time history, and any other dynamic analysis.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) or higher.

When a rigid floor diaphragm is present in the model, the program follows the following logic:

a. If a reference load type MASS is present, then the mass model is formed by combining all MASS reference loads.

b. If reference load type MASS is not present, then the mass model is formed by combining all gravity reference loads. Warnings are displayed in the analysis output to alert you to this fact.

c. If neither reference load types MASS nor gravity are present, then the mass model is formed by combining all DEAD reference loads. If any LIVE reference loads are present, then they will also be combined to this mass model. Warnings are displayed in the analysis output to alert you to this fact.

Mass model will be formed from Gravity or Dead/Live reference loads in case Mass reference load types are not present, only when rigid floor diaphragm is present in the model.

**General Format**

```
DEFINE REFERENCE LOADS

LOAD  Ri title
...
Load header and items
...
LOAD  Rj LOADTYPE MASS title
...
Load header and items
...
END DEFINE REFERENCE LOADS

DEFINE UBC/1893/... LOAD

ZONE ...

LOAD 1 title

UBC/1893/... LOAD X f1

LOAD 2 title

UBC/1893/... LOAD Z f2

LOAD 3 title

SPECTRUM ...
```
The mass model is defined using the LOADTYPE MASS command. This mass model will be used for all types of analysis, including static equivalent, response spectrum, time history, and simply Eigen solution. If the load type MASS is missing and also no mass is defined in the corresponding seismic or dynamic analysis load cases, the program will report error as mass is missing.

If a mass model using reference load type MASS is defined and also a seismic weight table is defined in DEFINE UBC/1893/… DEFINITION (or seismic mass is defined as part of a response spectrum or time history loading), the program will simply the later in place of the former for the seismic weight calculation. The program will issue a warning message in the analysis output. Care should be taken so that the mass model is not defined twice.

Example

UNIT FEET KIP
DEFINE REF LOAD
LOAD R1 LOADTYPE MASS
  * MASS MODEL
  SELFWEIGHT X 1
  SELFWEIGHT Y 1
  SELFWEIGHT Z 1
  JOINT LOAD
  17 TO 48 FX 2.5 FY 2.5 FZ 2.5
  49 TO 64 FX 1.25 FY 1.25 FZ 1.25
  *
LOAD R2
  SELFWEIGHT Y -1
  JOINT LOAD
  17 TO 48 FY -2.5
  49 TO 64 FY -1.25
END DEF REF LOAD
*************************************************************
DEFINE UBC LOAD
ZONE 0.38 I 1 RWX 5.6 RWZ 5.6 STYP 2 CT 0.032 NA 1.3 NV 1.6
*************************************************************
DEFINE TIME HISTORY
TYPE 1 FORCE
  0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
  TYPE 2 ACCELERATION
  0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
ARRIVAL TIMES
  0.0
DAMPING 0.05
*************************************************************
LOAD 1
  UBC LOAD X 0.75
*************************************************************
LOAD 2
  BC LOAD Z 0.75
*************************************************************
LOAD 3
  SPECTRUM CQC X 0.174075 ACC DAMP 0.05 SCALE 32.2
  0.03 1.00 ; 0.05 1.35 ; 0.1 1.95 ; 0.2 2.80 ; 0.5 2.80 ; 1.0 1.60
*************************************************************
LOAD 4
  SPECTRUM CQC Z 0.174075 ACC DAMP 0.05 SCALE 32.2
  0.03 1.00 ; 0.05 1.35 ; 0.1 1.95 ; 0.2 2.80 ; 0.5 2.80 ; 1.0 1.60
*************************************************************
TR.31.9 Defining Starting Load

This command is used to directly specify the starting load vectors for use with load dependant Ritz vector method for the eigen solution.

General Format

```
DEFINE STARTING LOAD
MASS  { X | Y | Z | XY | YZ | ZX } NGEN n
```

or

```
REFERENCE LOAD Ri NGEN n
```

END

You may specify one of each direction or direction pair when using the MASS option. You may specify any number of reference loads.

where

\[ n = \text{The number of Ritz vectors to be extracted corresponding to each starting load.} \]

\[ Ri = \text{The reference load number to be used as a starting load.} \]

Program Versus User-Generated Start Vectors

Using the program generated method, the starting load vector is generated using the mass model of the structure. Assuming mass matrix only has translational components, the resulting load vector will have force components in the directions of all the translational degrees of freedom (DOFs) and will have zero values for all the rotational DOFs. This will guarantee a static deflection with components in many directions to start with the initial mode assumption. As long as the initial mode assumption has a component in the direction of interest, it will yield a correct set of Ritz vectors.

With the user-defined starting load, the load vector will have a force component in one translational degree of freedom in the direction of the dynamic load. This results in a static deflection mainly in the direction of interest. In some models where mass participation is predominant in one translational direction for the initial few modes this method can achieve 90% mass participation with only few modes.

Note: A single set of Ritz vectors can be derived from a single starting load. If all starting loads that participate in dynamic loads are defined, a single set of Ritz vectors can be extracted from all participating starting loads. The STAAD.Pro Advanced Analysis can extract a single set of Ritz vector only from one single starting load. Thus, the use of user-defined start vectors is best suited if the structural response is predominant in one translational direction. If responses in multiple translational DOF are predominant, it is recommended to use the program generated starting load vector.
Notes
Use of this command (and specifying the eigen solution in general) requires the Advanced Analysis.

This command requires that the \texttt{SET \ EIGEN \ METHOD \ RITZ} command also be used previously in the input file. Refer to \textit{TR.5 Set Command Specification} (on page 2413) for details.

\begin{verbatim}
Examples
The following uses starting loads in each direction, with 10 Ritz vectors in each except for the Y direction only, which will extract 5.

\texttt{DEFINE \ STARTING \ LOAD}
* Load components in X direction only
\texttt{MASS X NGEN 10}
* Load components in Y direction only
\texttt{MASS Y NGEN 5}
* Load components in Z direction only
\texttt{MASS Z NGEN 10}
* Load components in X & Y directions
\texttt{MASS XY NGEN 10}
* Load components in Y & Z directions
\texttt{MASS YZ NGEN 10}
* Load components in Z & X directions
\texttt{MASS ZX NGEN 10}
\texttt{END}

The following example contains four reference loads:
1. \texttt{R1}: all 3 translational degrees of freedom (X, Y, & Z)
2. \texttt{R2}: in X d.o.f. only
3. \texttt{R3}: in Y d.o.f. only
4. \texttt{R4}: in Z d.o.f. only

Twenty five Ritz vectors will be generated for each of the starting loads.

\texttt{DEFINE \ STARTING \ LOAD}
* Load components in all directions
\texttt{REF \ LOAD \ R1 \ NGEN 25}
* Load components in X direction only
\texttt{REF \ LOAD \ R2 \ NGEN 25}
* Load components in Y direction only
\texttt{REF \ LOAD \ R3 \ NGEN 25}
* Load components in Z direction only
\texttt{REF \ LOAD \ R4 \ NGEN 25}
\texttt{END}
\end{verbatim}

Related Links
- \textit{M. To use starting vectors with load-dependant Ritz vectors} (on page 865)
- \textit{G.17.3.1 Solution of the Eigenproblem} (on page 2363)
- \textit{M. To use starting vectors with load-dependant Ritz vectors} (on page 865)
TR.32 Loading Specifications

This section describes the various loading options available in STAAD.Pro. The following command may be used to initiate a new load case.

**General Format**

```
LOADING i1 ( LOADTYPE a1 ) ( REDUCIBLE ) ( TITLE any_load_title )
```

Where:
- **LOADING i1** any unique integer number (up to five digits) to identify the load case. This number need not be sequential with the previous load number.
- **LOADTYPE a1** one of the following:

<table>
<thead>
<tr>
<th>Load Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dead</td>
<td></td>
</tr>
<tr>
<td>Rain Water/ Ice</td>
<td></td>
</tr>
<tr>
<td>Wind on Ice</td>
<td></td>
</tr>
<tr>
<td>Live</td>
<td></td>
</tr>
<tr>
<td>Ponding</td>
<td></td>
</tr>
<tr>
<td>Crane Hook</td>
<td></td>
</tr>
<tr>
<td>Roof Live</td>
<td></td>
</tr>
<tr>
<td>Dust</td>
<td></td>
</tr>
<tr>
<td>Mass (See Notes on page 2650)</td>
<td></td>
</tr>
<tr>
<td>Wind</td>
<td></td>
</tr>
<tr>
<td>Traffic</td>
<td></td>
</tr>
<tr>
<td>Gravity</td>
<td></td>
</tr>
<tr>
<td>Seismic</td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td></td>
</tr>
<tr>
<td>Push</td>
<td></td>
</tr>
<tr>
<td>Snow</td>
<td></td>
</tr>
<tr>
<td>Accidental</td>
<td></td>
</tr>
<tr>
<td>None</td>
<td></td>
</tr>
<tr>
<td>Fluids</td>
<td></td>
</tr>
<tr>
<td>Flood</td>
<td></td>
</tr>
<tr>
<td>Soil</td>
<td>Ice</td>
</tr>
</tbody>
</table>

The words **LOADTYPE a** are necessary only if you intend to use the Automatic Load Combination generation tool in the graphic interface. For details, refer to the Auto Load Combination dialog (on page 3033).

The keyword **REDUCIBLE** should be used only when the loadtype is **LIVE**. It instructs the program to reduce – according to the provisions of UBC 1997, IBC 2000, or IBC 2003 codes – a floor live load specified using **FLOOR LOAD** or **ONEWAY LOAD** commands. For details, see TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664).

Under this heading, all different loads related to this loading number can be input. These different kinds of loads are described in the remaining sub-sections below.

**Notes**

a. For Mass Model Loading in Dynamics, it is strongly recommended that you read G.17.3.2 Mass Modeling (on page 2365) and TR.31.8.3 Mass Model Using Reference Load (on page 2646). For the purpose of entering the mass distribution for the first dynamic load case, use the following sections:

  TR.32.1 Joint Load Specification (on page 2651)
A reference load can be used to create a mass model for all dynamic analyses.

The purpose of the mass modeling step is to create lumped masses at the joints that the eigensolution can use. The member/element loading is only a convenience in generating the joint masses. Analytically the masses are not in the elements but are lumped at the joints.

b. The absolute value of joint loads or loads distributed to joints from member/element loadings will be treated as weights. The moments applied to member/elements or computed at joints as a result of member/element loadings will be ignored. Only moments (actually weight moment of inertia, force-length\(^2\) units) applied in the joint.

c. The load command will be used in defining the weight moment of inertias at joints. For slave joint directions, the associated joint weight or weight moment of inertia will be moved to the master. In addition, the translational weights at slave joint directions will be multiplied by the square of the distance to the master to get the additional weight moment of inertia at the master. Cross-product weight moment of inertias at the master will be ignored.

Related Links
• M. To create a new primary load case (on page 822)
• M. To define primary load type (on page 867)
• G.17.3.2 Mass Modeling (on page 2365)

TR.32.1 Joint Load Specification

This set of commands may be used to specify joint loads on the structure. For dynamic mass modeling see TR.32.10 Dynamic Loading Specification (on page 2686) and G.17.3 Dynamic Analysis (on page 2362).

General Format

```
JOINT LOAD
joint-list (inclined-spec) *{ FX f8   | FY f9   | FZ f10 | MX f11 | MY f12 | MZ f13 }
```

Where:
```
inclined-spec = INCLINED { f1 f2 f3 | REF f4 f5 f6 | REFJT f7 }
```

Use the optional \textit{inclined-spec} to specify a joint load in a inclined load axis. See Inclined Load Axis System (on page 2652) below for details.

- \(f1, f2, f3\) \(x, y, z\) relative distances from the joint in the global directions to the reference point
- \(f4, f5, f6\) \(x, y, z\) global coordinates of an arbitrary reference point
- \(f7\) a joint number whose \(x, y, z\) global coordinates are the reference point
- \(f8, f9, f10\) force values in the corresponding global direction (even at inclined support joints).
- \(f11, f12, f13\) moment values in the corresponding global direction.
Example

UNIT FEET KIP
* ...
JOINT LOAD
3 TO 7 9 11 FY -17.2 MZ 180.0
5 8 FX 15.1
UNIT INCH KIP
12 MX 180.0 FZ 6.3

Notes
a. Joint numbers may be repeated where loads are meant to be additive in the joint.
b. A UNIT command may be on lines in between joint-list lines.
c. If moments are for dynamic mass, then the units are assumed to be force-length².

Inclined Load Axis System

If the joint loading is specified using the option INCLINED, then the axes in which the direct forces (and about which for moment forces) are applied are realigned from the global directions of X, Y, and Z to the directions x', y', and z' as shown below.

The reference point –whether it is specified using absolute coordinates, relative distances, or a reference joint– defines the direction of the x' axes from the loaded node to the reference point. The direction of the Y' axes is then defined as the direction perpendicular to x' that lies in the plane of X' and Y. In the special case of when X' is in the direction of Y, Y' is taken to be the direction of Z. The direction of Z' is then defined as the direction perpendicular to the plane X' and Y' and follows the right-hand rule as for all other axes systems used in STAAD.Pro.
Example

JOINT LOAD
4 INCLINED 1.0 -1.0 0.0 FX 10

JOINT LOAD
3 INCLINED REFJT 7 FZ -30

Related Links
- G.15.1 Joint Loads (on page 2337)
- M. To add a nodal load (on page 824)
- G.15.1 Joint Loads (on page 2337)

TR.32.2 Member Load Specification

This set of commands may be used to specify member loads on frame members

General Format

```
MEMBER LOAD
member-list { { UNI | UMOM } dir-spec f_1 f_2 f_3 f_4 | { CON | CMOM } dir-spec f_5 f_6 f_4 | LIN dir-spec f_7 f_8 f_9 | TRAP dir-spec f_10 f_11 f_12 f_13 }
```

Where:

```
dir-spec = { X | Y | Z | GX | GY | GZ | PX | PY | PZ }
```

- X, Y, & Z specify the direction of the load in the local (member) x, y and z-axes.
- GX, GY, & GZ specify the direction of the load in the global X, Y, and Z-axes.
- PX, PY, & PZ may be used if the load is to be along the projected length of the member in the corresponding global direction.

**Note:** Load start and end distances are measured along the member length and **not** the projected length.

- **f_1** value of uniformly distributed load (UNI) or moment (UMOM).
- **f_2, f_3** distance of from the start of the member to the start of the load, and distance from the start of the member to the end of the load, respectively. The load is assumed to cover the full member length if f_2 and f_3 are omitted.
Note: Uniformly distributed moments cannot be assigned to tapered members for analysis.

**f4**  
Perpendicular distance from the member shear center to the local plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis. If global or projected load is selected, then the local Y component of load is offset the \( f_4 \) distance; the local Z component is offset the \( f_4 \) distance; and the local X component is not offset.

Note: The local x component of force is not offset (i.e., no secondary moment is caused by axial load).

**f5**  
Value of concentrated force (CON) or moment (CMOM)

**f6**  
Distance of from the start of the member to concentrated force or moment. \( f_6 \) will default to half the member length if omitted.

**f7, f8**  
LIN specifies a linearly decreasing or increasing, or a triangular load. If the load is linearly increasing or decreasing then \( f_7 \) is the value at the start of the member and \( f_8 \) is the value at the end.

**f9**  
If the load is triangular, then \( f_7 \) and \( f_8 \) are input as zero and \( f_9 \) is the value of the load in the middle of the member.

**f10, f11**  
The starting and ending load value for a trapezoidal linearly varying load (TRAP), respectively. The TRAP load may act over the full or partial length of a member and in a local, global or projected direction.

**f12, f13**  
The loading starting point and stopping point, respectively. Both are measured from the start of the member. If \( f_{12} \) and \( f_{13} \) are not given, the load is assumed to cover the full member length.
Notes

a. In earlier versions of STAAD, the LINear type of member load could be applied only along the local axis of the member. It has been modified to allow for global and projected axes directions also.

b. If the member being loaded has offset distances (see MEMBER OFFSET specification), the location of the load is measured not from the coordinates of the starting node but from the offset distance.

c. Trapezoidal loads are converted into a uniform load and 8 or more concentrated loads.

d. A UNIT command may be on lines in between member-list lines.

e. If a load location is less than zero, it is reset to 0.0.

f. If a load location is greater than the length, it is reset to the length.

Example

MEMBER LOAD
619 CON GY -2.35 5.827
68 TO 72 UNI GX -0.088 3.17 10.0
186 TRAP GY -0.24 -0.35 0.0 7.96
3212 LIN X -5.431 -3.335
41016 UNI PZ -0.075
3724 LIN GY -6.2 -7.8

Related Links

- G.15.2 Member Load (on page 2337)
- M. To add a concentrated force or moment on members (on page 826)
- M. To add a uniform load to members (on page 826)
- M. To add a linear varying load to members (on page 827)
- M. To add a hydrostatic load to objects (on page 833)
- G.15.2 Member Load (on page 2337)

TR.32.2.1 PMember Load Specification

This set of commands may be used to specify member loads on a physical member (pmember).

General Format

PMEMBER LOAD

pmember-list { { UNI | UMOM } dir-spec f1 f2 f3 f4 | { CON | CMOM } dir-spec f5 f6 f4 | TRAP dir-spec f7 f8 f9 f10 }

Where:

dir-spec = { X | Y | Z | GX | GY | GZ | PX | PY | PZ }

X, Y, & Z specify the direction of the load in the local (member) x, y and z-axes.
GX, GY, & GZ specify the direction of the load in the global X, Y, and Z-axes.
PX, PY, & PZ may be used if the load is to be along the projected length of the member in the corresponding global direction.

Note: Load start and end distances are measured along the member length and not the projected length.
**Technical Reference of STAAD Commands**

**TR.32 Loading Specifications**

\[ f_1 \] value of uniformly distributed load (UNI) or moment (UMOM).

\[ f_2, f_3 \] distance of from the start of the member to the start of the load, and distance from the start of the member to the end of the load, respectively. The load is assumed to cover the full member length if \( f_2 \) and \( f_3 \) are omitted.

![Diagram showing \( f_2, f_3, f_4 \) values](image)

**Note:** Uniformly distributed moments can not be assigned to tapered members for analysis.

\[ f_4 \] Perpendicular distance from the member shear center to the local plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis. If global or projected load is selected, then the local Y component of load is offset the \( f_4 \) distance; the local Z component is offset the \( f_4 \) distance; and the local X component is not offset.

![Diagram showing \( f_4 \) value](image)

**Note:** The local x component of force is not offset (i.e., no secondary moment is caused by axial load).

\[ f_5 \] value of concentrated force (CON) or moment (CMOM)

\[ f_6 \] distance of from the start of the member to concentrated force or moment. \( f_6 \) will default to half the member length if omitted.

\[ f_7, f_8 \] The starting and ending load value for a trapezoidal linearly varying load (TRAP), respectively. The TRAP load may act over the full or partial length of a member and in a local, global or projected direction.

\[ f_9, f_{10} \] the loading starting point and stopping point, respectively. Both are measured from the start of the member. If \( f_9 \) and \( f_{10} \) are not given, the load is assumed to cover the full member length.
TR.32.3 Element Load Specifications

This set of commands may be used to specify various types of loads on plate and solid elements.

Related Links
- G.15.8 Loading on Elements (on page 2342)
- G.5 Finite Element Information (on page 2308)
- G.5.1 Plate and Shell Elements (on page 2308)
- G.5.2 Solid Elements (on page 2319)
- G.5.3 Surface Elements (Deprecated) (on page 2322)

TR.32.3.1 Element Load Specification - Plates

This command may be used to specify various types of ELEMENT LOADS for plates.

Plate element loads must be applied following the expression

```
ELEMENT LOAD (PLATE)
```

using the format explained under the following options.

Option 1

```
element-list {PRESSURE {GX | GY | GZ} p1 (x1 y1 x2 y2) }
```

This is for specifying a pressure of magnitude \( p1 \) in one of the global axis directions on the full element or a small rectangular part of an element. If applied on a small part, \( (x1,y1,x2 \text{ and } y2) \) define the corners of the rectangular region where the load is applied. If only \( x1, y1 \) is provided, the load is assumed as a concentrated load applied at the specified point defined by \( (x1,y1) \). If \( (x1,y1,x2,y2) \) is not provided, the load is assumed to act over the full area of the element. \( (x1,y1,x2 \text{ and } y2) \) are measured from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

\( p1 \) has units of force per square of length for pressure and units of force for concentrated load.

\( GX, GY \) and \( GZ \) represent the global axis directions.

Option 2

```
element-list PRESSURE p1 (x1 y1 x2 y2)
```

This is for specifying a constant pressure of magnitude \( p1 \) acting perpendicular to the plane of the element on the full element or a small rectangular part of an element. This coincides with the element’s local Z axis. If applied on a small part, \( (x1,y1,x2 \text{ and } y2) \) define the corners of the rectangular region where the load is applied. If only \( x1, y1 \) is provided, the load is assumed as a concentrated load applied at the specified point defined by \( (x1,y1) \). If \( (x1,y1,x2,y2) \) is not provided, the load is assumed to act over the full area of the element. \( (x1,y1, x2 \text{ and } y2) \) are measure from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

\( p1 \) has units of force per square of length for pressure and units of force for concentrated load.
Figure 308: Coordinate values, x1,y1 & x2,y2, in the local coordinate system used in options 1 and 2

Option 3

`element-list PRESSURE {LX | LY} {p2}`

This is for specifying a constant pressure of magnitude p2 along the local X (LX) or Y (LY) axis of the element (parallel to the element surface). An example of this type of load is friction load. With this option, a load can be applied only on the full area of the element.

p2 has units of force per square of length.

Option 4

`element-list TRAP {GX | GY | GZ | LX | LY} {X | Y} f1 f2`

This is for applying a trapezoidally varying load with the following characteristics:

a. The direction of action of the load is global (GX, GY or GZ), parallel to the surface (LX or LY as in friction type loads), or normal to the element surface (local Z). The last becomes the automatic direction of action if the global or tangential directions are not specified.

b. The load varies along the local X or Y directions (imagine the wall of a tank with hydrostatic pressure where pressure at the lower nodes is higher than at the upper nodes, and hence the load varies as one travels from the bottom edge of the elements to the top edge.)

This type of load has to be applied over the full area of the element. f1 is the intensity at the I-J (or J-K) edge, and f2 is the intensity at the K-L (or L-I) edge depending on whether the load varies along "X" or "Y".

The TRAP option should be used when a linearly varying pressure needs to be specified. The variation must be provided over the entire element.

X or Y - Direction of variation of element pressure. The TRAP X/Y option indicates that the variation of the Trapezoid is in the local X or in the local Y direction. The load acts in the global or local direction if selected, otherwise along the local Z axis.

- \( f_1 \) - Pressure intensity at start.
- \( f_2 \) - Pressure intensity at end.

**Option 5**

```
\text{element-list} \ \text{TRAP} \ \{\text{GX} | \text{GY} | \text{GZ} | \text{LX} | \text{LY}\} \ \text{JT} \ f_3 \ f_4 \ f_5 \ f_6
```

This is for specifying a trapezoidally varying load over the full area of the element where one happens to know the intensity at the joints (JT) of the element. The load is defined by intensities of \( f_3, f_4, f_5 \) and \( f_6 \) at the 4 corners of a 4-noded element. For triangular elements, \( f_6 \) is not applicable. The load can act along the global directions (GX, GY and GZ) or along the local X and Y directions (LX and LY, like a friction load).

**Notes**

- a. “Start” and “end” defined above are based on positive directions of the local X or Y axis.
- b. Pressure intensities at the joints allows linear variation of pressure in both the X and Y local directions simultaneously.
- c. The TRAP load with global directions may be used to apply a volumetric type of pressure. For example, consider a grain silo with a sloping wall. In the event of modeling it using non-uniform elements, by which we mean elements whose 3 or 4 nodes are all at different elevations, the grain height at each node will depend on the elevation of the node. One can apply the pressure by specifying the intensity at each node of each element.
Example

LOAD 4
ELEMENT LOAD
  1 7 TO 10 PR 2.5
  11 12 PR 2.5 1.5 2.5 5.5 4.5
  15 TO 25 TRAP X 1.5 4.5
  15 TO 20 TRAP GY JT 1.5 4.5 2.5 5.5
  34 PR 5.0 2.5 2.5
  35 TO 45 PR -2.5
  15 25 TRAP GX Y 1.5 4.5
  29 95 TRAP LX Y 3.7 8.7

Related Links

- G.15.8 Loading on Elements (on page 2342)
- M. To add pressure load on a plate (on page 829)
- G.5 Finite Element Information (on page 2308)
- G.5.1 Plate and Shell Elements (on page 2308)
- G.5.2 Solid Elements (on page 2319)
- G.5.3 Surface Elements (Deprecated) (on page 2322)

TR.32.3.2 Element Load Specification - Solids

The following types of loads can be assigned on the individual faces of solid elements:

- A uniform pressure
- A volumetric type of pressure on a face where the intensity at one node of the face can be different from that at another node on the same face.

An example of such a load is the weight of water on the sloping face of a dam. If the dam is modeled using solids, for the individual elements, the water height at the lower elevation nodes will be larger than those at the higher elevation nodes.

General Format

The syntax is as follows.

```
ELEMENT LOAD SOLIDS
elem-list FACE i1 PRESSURE { GX | GY | GZ } f1 f2 f3 f4
```

Where:

- \( f_1 f_2 f_3 f_4 \) = Pressure values at the joints for each 3 or 4 joint face defined. Only \( f_1 \) needs to be specified for uniform pressure. In any case the pressure is provided over the entire face.
- \( i_1 \) = one of six face numbers to receive the pressure for the solids selected. See the figure in G.5.2 Solid Elements (on page 2319) for the following face definitions. Enter a pressure on all 4 joints even if the face is collapsed to 3 points.

Description

The first option above loads the solid by specifying one or more of the 6 faces to receive pressure.
The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be normal to the face and a positive pressure is into the solid. The loads are proportional to the area, not the projected area.

<table>
<thead>
<tr>
<th>Face Number</th>
<th>Surface Joints</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>f1</td>
</tr>
<tr>
<td>1 front</td>
<td>Jt 1</td>
</tr>
<tr>
<td>2 bottom</td>
<td>Jt 1</td>
</tr>
<tr>
<td>3 left</td>
<td>Jt 1</td>
</tr>
<tr>
<td>4 top</td>
<td>Jt 4</td>
</tr>
<tr>
<td>5 right</td>
<td>Jt 2</td>
</tr>
<tr>
<td>6 back</td>
<td>Jt 5</td>
</tr>
</tbody>
</table>

Example

LOAD 4
ELEMENT LOAD SOLIDS
11 12 FACE 3 PR 2.5 1.5 2.5 5.5

Related Links
- [G.5.2 Solid Elements](on page 2319)

TR.32.3.3 Element Load Specification - Joints

This command may be used to specify various types of element like loads for joints. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads. This command may be used as an alternative or supplement for the Area Load, Floor Load, and the other Element Load commands.

The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be in the local Z direction as if the joints defined a plate. The loads are proportional to the area, not the projected area.

General Format

```
ELEMENT LOAD JOINTS

I1 (BY I2) I3 (BY I4) I5 (BY I6) I7 (BY I8) -
FACETS J1 PRESSURE { GX | GY | GZ } F1 F2 F3 F4
```

**Note:** If this data is on more than one line, the hyphens must be within the joint data.

Where:
$f_1 f_2 f_3 f_4 = \text{Pressure values at the joints for each 3 or 4 joint facet defined. Only $f_1$ needs to be specified for uniform pressure. In any case the pressure is provided over the entire element.}$

$j_1 = \text{number of facets loaded.}$

$i_1, i_3, i_5, i_7 = \text{Joint numbers that define the first facet.}$

$i_2, i_4, i_6, i_8 = \text{each joint number is incremented by the BY value (1 if omitted).}$

### Example

```plaintext
LOAD 4
ELEMENT LOAD JOINT
1 BY 1 2 BY 1 32 BY 1 31 BY 1 -
FACETS 5 PR GY 10 10 15 15
```

The above data is equivalent to the following:

```plaintext
LOAD 4
ELEMENT LOAD JOINT
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
2 3 33 32 FACETS 1 PRESSURE GY 10 10 15 15
3 4 34 33 FACETS 1 PRESSURE GY 10 10 15 15
4 5 35 34 FACETS 1 PRESSURE GY 10 10 15 15
5 6 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

So, the value following the word FACETS is like a counter for generation, indicating how many element faces the load command must be created for. Thus a value of 5 for facets means, a total of 5 imaginary element faces have been loaded.

**BY** is the value by which the individual corner node number is being incremented during the generation. In this example, the value is 1, which is same as the default. Instead, if it had been say,

```plaintext
1 BY 22 BY 2 32 BY 1 31 BY 1 -
FACETS 5 PRESSURE GY 10 10 15 15
```

we would have obtained

```plaintext
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
3 5 33 32 FACETS 1 PRESSURE GY 10 10 15 15
5 8 34 33 FACETS 1 PRESSURE GY 10 10 15 15
8 11 35 34 FACETS 1 PRESSURE GY 10 10 15 15
9 14 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

### Notes

If a pressure or volumetric load is acting on a region or surface, and the entity which makes up the surface, like a slab, is not part of the structural model, one can apply the pressure load using this facility. The load is defined in terms of the pressure intensity at the 3 or 4 nodes which can be treated as the corners of the triangular or quadrilateral plane area which makes up the region. This command may be used as an alternative or supplement for the Area Load, Floor Load, Wind Load, and other pressure load situations.

In other words, the element pressure load can be applied along a global direction on any surface, without actually having elements to model that surface. Thus, for a sloping face of a building, if one wants to apply a wind pressure on the sloping face, one can do so by specifying the joints which make up the boundary of that face. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads.
TR.32.3.4 Surface Loads Specification

The following loading options are available for surface entities:

**Note:** Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

### General Format

```
LOAD n
SURFACE LOAD

surface-list { PRESSURE | TRAPEZOIDAL } ( { GX | GY | GZ } ) load-spec
```

or

```
SSELFWT { X | Y | Z } f9 LIST surface-list
```

Where:

- GX, GY, and GZ = Global X, Y and Z directions. If the direction is omitted, the load will act along the local Z axis of the surface.

`load-spec` is provided for the following types of surface loads:

- **Uniform pressure across the entire surface. Use with PRE command.**
  
  ```
  Load-spec = w
  ```

  Where:

  - `w` = Value of pressure. Negative value indicates load acts opposite to the direction of the axis specified.

- **Pressure load on a partial area of the surface. Use with PRE command.**
  
  ```
  Load-spec = w x1 y1 x2 y2
  ```

  Where:

  - `w` = Value of pressure. Negative value indicates load acts opposite to the direction of the axis specified.
  - `x1, y1` = Local X and Y coordinates of the corner nearest to the surface origin of the loaded region.
  - `x2, y2` = Local X and Y coordinates of the corner farthest from the surface origin of the loaded region.

- **Concentrated force anywhere on the surface. Use with PRE command.**
  
  ```
  Load-spec = p x y
  ```

  Where:

  - `p` = Value of the concentrated force. Negative sign indicates load acts opposite to the direction of the axis specified.
  - `x, y` = Local X and Y coordinates of the point of action of the force. Measured from the origin of the surface, in the local coordinate system of the surface.

- **Trapezoidal pressure load on a partial area of the surface. Use with TRAP command.**
  
  ```
  Load-spec = f1 f2 f3 f4 CORNER f5 f6 f7 f8
  ```
Where:

$f_1, f_2, f_3, f_4$ = the pressure at the corners of the loaded area. Negative values indicate loads act opposite to the direction of the axis specified.

$f_5, f_6$ = Local X and Y coordinates of the first corner of the loaded area, respectively.

$f_7, f_8$ = Local X and Y coordinates of the second corner of the loaded area, respectively. These values must be different than the corresponding values of the first corner so that the loaded area is rectangular.

$f_9$ = Magnification factor for surface selfweight. Negative value indicates load acts opposite to the direction of the axis specified.

**Example**

LOAD 1 Uniform Pressure on Full Surface
SURFACE LOAD
1 2 PRESS GX 0.002
3 GX -0.0025
*
LOAD 2 Concentrated Force
SURFACE LOAD
10 12 PRE GZ 400 3.5 4.5
*
LOAD 3 Partial Area Load
SURFACE LOAD
23 25 PRE FY -250 4 4.3 8 9.5
*
LOAD 4 Partial Surface Trapezoidal Load
SURFACE LOAD
11 13 TRAP GY -120 -240 -240 -360 COR 2.5 6.5 3.5 4.5
*
LOAD 5 Surface Selfweight
SSELFWT Y -1 LIST ALL

**TR.32.4 Area, One-way, and Floor Load Specifications**

These commands may be used to specify loading over an area enclosed by beam members and the program will distribute the loading onto the perimeter beams as either a one way or two-way system. They are used mostly when the entity transmitting the load, such as a slab, is not part of the structural model. The AREA LOAD or ONEWAY LOAD may be used for modeling one-way distribution and the FLOOR LOAD may be used for modeling two-way distribution. There are three commands which should be used in the following way:

**Area Load**
The program will establish the direction of the shorter span and load the beams in that direction.

**Floor Load**
This command is used for distributing a pressure load onto to the beams that define a closed loop.

**One-way**
This may appear similar to the AREA LOAD command, but this command defines an area from which the program will search out closed loops of beams, similar to the FLOOR LOAD command and also has the option to define the direction of span. This command is an development of the principals defined in the FLOOR LOAD command, but the load is defined to span in a single direction.
Live Load Reduction

Floor loads and one-way loads can be made reducible according to Section 1607 of IBC 2000 if included in a Live Load case which is specified as Reducible. The following rules are implemented:

- Reduction only applies to live loads on members with a tributary area of greater than 150 ft\(^2\).
- Live loads over 100 psf are not reduced.
- The reduction in live load (in percent) is \(0.08 \times (A_{trib} - 150)\), with a maximum reduction amount of 40% for members carrying load from one level only.

**Note:** Live load reduction is **not** not available for area loads.

Related Links

- G.16.2 Seismic Load Generator (on page 2343)
- G.15.3 Area, One-way, and Floor Loads (on page 2338)

TR.32.4.1 Area Load Specification

Used for distributing a pressure load onto the beams that define a closed loop. The program will establish the direction of the shorter span and load the beams in that direction.

**Note:** The AREA LOAD command has been deprecated in favor of the ONEWAY LOAD or FLOOR LOAD commands.

General Format

```
AREA LOAD
memb-list ALOAD f1 { GX | GY | GZ }
```

Where:

f1 The value of the area load (units of weight over square length unit). If Global direction is omitted, then this load acts along the positive local y-axis (for the members of a floor analysis, the local y direction will coincide with global vertical axis in most cases). If Global direction is included, then the load acts in that direction. The magnitude of the loads calculated is the same as if the positive local y axis option was selected. (For detailed description, refer to G.15.3 Area, One-way, and Floor Loads (on page 2338).)

**Note:** Area load should not be specified on members declared as MEMBER CABLE, MEMBER TRUSS or MEMBER TENSION.

Example

```
AREA LOAD
2 4 TO 8 A LOAD - 0.250
12 16 ALOAD -.0500
```

Notes

a. The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).
b. For the **FLOOR LOAD** specification, a two-way distribution of the load is considered. For the **ONEWAY** and **AREA LOAD** specification, a one-way action is considered. For **ONEWAY** loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the **FLOOR LOAD** type.

c. The global horizontal direction options (GX and GZ) enables one to consider **AREA LOADs**, **ONEWAY LOADs**, and **FLOOR LOADs** for mass matrix for frequency calculations.

**Related Links**
- G.15.3 Area, One-way, and Floor Loads (on page 2338)
- M. To add an area load (on page 830)
- G.15.3 Area, One-way, and Floor Loads (on page 2338)

**TR.32.4.2 One-way Load Specification**

Defines an area from which the program will search out closed loops of beams, similar to the **FLOOR LOAD** (on page 2672) command and also has the option to define the direction of span. This command is a development of the principals defined in the **FLOOR LOAD** command, but the load is defined to span in a single direction (similar to **AREA LOAD**).

The One-way Load specification be applied to groups and can also use live load reduction per IBC or UBC codes.

**General Format**

```
ONeway LOAD (SAVE { LOAD })

Yrange \( f_1 \ f_2 \) oneLOAD \( f_3 \) (XRA \( f_4 \ f_5 \ ZRA \ f_6 \ f_7 \) \{ GX | GY | GZ \} (TOWARDS \( f_8 \)) (PRINT \{ MEMBER | LOAD \})
```

or
```
Xrange \( f_1 \ f_2 \) oneLOAD \( f_3 \) (YRA \( f_4 \ f_5 \ ZRA \ f_6 \ f_7 \) \{ GX | GY | GZ \} (TOWARDS \( f_8 \)) (PRINT \{ MEMBER | LOAD \})
```

or
```
Zrange \( f_1 \ f_2 \) oneLOAD \( f_3 \) (XRA \( f_4 \ f_5 \ YRA \ f_6 \ f_7 \) \{ GX | GY | GZ \} (TOWARDS \( f_8 \)) (PRINT \{ MEMBER | LOAD \})
```

or
```
_FloorGroupName \( f_3 \) \{ GX | GY | GZ \} (INCLINED) (TOWARDS \( f_8 \))
```

Where:
- \( f_1 \ f_2 \) Global coordinate values to specify Y, X, or Z range. The load will be calculated for all members lying in that global plane within the first specified global coordinate range.
- \( f_3 \) The value of the load (unit weight over square length unit). If the global direction is omitted, then this load acts parallel to the positive global Y if command begins with YRA and based on the area projected on a X-Z plane. Similarly, for commands beginning with XRA, the load acts parallel to the positive global X and based on the area projected on a Y-Z plane. Similarly, for commands beginning with ZRA, the load acts parallel to the positive global Z and based on the area projected on a X-Y plane.
- \( f_4-f_7 \) Global coordinate values to define the corner points of the area on which the specified floor load \( f_3 \) acts. If not specified, the floor load will be calculated for all members in all floors within the first specified global coordinate range.
GX,GY,GZ If a Global direction is included, then the load is re-directed to act in the specified
direction(s) with a magnitude of the loads which is based on the area projected on a plane
as if the Global direction was omitted. The Global direction option is especially useful in
mass definition.

FloorGroupName See TR.16.1 Listing of Entities by Specifying Groups (on page 2440) for the procedure for
creating FLOORGROUPs. The member-list contained in this name will be the candidates that
will receive the load generated from the floor pressure.

f8 Defines a member onto which the loading is directed and defines the span direction for the
one way loading. If the TOWARDS option is not used, the program will default to distributing
a one-way load to the longest side. See Note b below for square panels.

INCLINED - This option must be used when a ONEWAY LOAD is applied on a set of members that form a panel(s)
which is inclined to the global XY, YZ, or ZX planes.

PRINT = This option is used when floor panel information printout is required. The total number of panels
identified, total area of all panels and total load generated will be printed in the output file.

PRINT MEMBER = This option is used to print panel member numbers on which loads are generated along with
the total number of panels identified, total area of all panels and total load generated.

PRINT LOAD = This option is used to print panel loading information. All other information available with PRINT
MEMBER will also be printed in the output file.

SAVE = This option will give panel information in an external text file named (filename)_FLD.TXT. The
information corresponding to PRINT MEMBER will be printed in this file.

SAVE LOAD = This option will give panel information in an external text file named (filename)_FLD.TXT. The
information corresponding to PRINT LOAD will be printed in this file.

Note: It is recommended to use SAVE or SAVE LOAD option which will give same output printed in an external
text file. If detailed output is printed in output (.ANL file) file using PRINT MEMBER or PRINT LOAD option, the
size of the output file may increase to a large extent.

Tip: The SET FLOOR LOAD TOLERANCE command may be used to specify the tolerance for out-of-plane
nodes to be included in a floor load. See TR.5 Set Command Specification (on page 2413)

Notes

a. The structure has to be modeled in such a way that the specified global axis remains perpendicular to the
floor plane(s).

b. For the FLOOR LOAD specification, a two-way distribution of the load is considered. For the ONEWAY and AREA
LOAD specification, a one-way action is considered. For ONE WAY loads, the program attempts to find the
shorter direction within panels for load generation purposes.

In V8i SELECTseries 6 build 20.07.11.50 and later: for one-way loads applied to square panels (i.e., span
direction is the same in either direction), triangular loads are generated on the members forming the four
sides of the square. The intensity of the loads, which are equal for all four members, is calculated on the basis
of four triangles formed by the two intersecting diagonals of the square.

Alternately, you may use the FLOOR LOAD type.

c. The load per unit area may not vary for a particular panel and it is assumed to be continuous and without
holes.
d. If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better.

The attached example illustrates a case where the floor has to be sub-divided into smaller regions for the floor load generation to yield proper results. The internal angle at node 6 between the sides 108 and 111 exceeds 180 degrees. A similar situation exists at node 7 also. As a result, the following command:

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35
```

will not yield acceptable results. Instead, the region should be subdivided as shown in the following example

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35 XRA -.01 15.1 ZRA -0.1 8.1
YRANGE 11.9 12.1 FLOAD -0.35 XRA 4.9 10.1 ZRA 7.9 16.1
```

![Diagram of floor with nodes and labels]


e. At least one quadrilateral panel bounded on at least 3 sides by "complete" members has to be present within the bounds of the user-defined range of coordinates (XRANGE, YRANGE and ZRANGE) in order for the program to successfully generate member loads from the FLOOR/ONEWAY LOAD specification. A "complete" member is defined as one whose entire length between its start and end coordinates borders the specified panel.

The load distribution pattern depends upon the shape of the panel. If the panel is Rectangular, the distribution will be Trapezoidal and triangular as explained in the following diagram.
First, the CG of the polygon is calculated. Then, each corner is connected to the CG to form triangles as shown. For each triangle, a vertical line is drawn from the CG to the opposite side. If the point of intersection of the vertical line and the side falls outside the triangle, the area of that triangle will be calculated and an equivalent uniform distributed load will be applied on that side. Otherwise a triangular load will be applied on the side.

**Live Load Reduction per UBC and IBC Codes**

The UBC 1997, IBC 2000, and IBC 2003 codes permit reduction of floor live loads under certain situations. The provisions of these codes have been incorporated in the manner described further below.

To utilize this facility, the following conditions have to be met when creating the STAAD.Pro model.

1. The live load must be applied using the `FLOOR` LOAD or `ONEWAY` LOAD option. This option is described above, and an example of its usage may be found in [EX. US-15 Wind and Floor Load Generation on a Space Frame](on page 4480).
ii. As shown in TR.32 Loading Specifications (on page 2650), the load case has to be assigned a Type called Live at the time of creation of that case. Additionally, the option called Reducible, also has to be specified as shown.

```
LOAD n LOADTYPE Live REDUCIBLE
```

Where:

\( n \) is the load case number

The following figures show the load generated on members for the two situations.

Figure 311: Load generated for the previously described cases
Table 255: Details of the code implementation

<table>
<thead>
<tr>
<th>Code Name</th>
<th>Section of code which has been implemented</th>
<th>Applicable Equations</th>
</tr>
</thead>
<tbody>
<tr>
<td>UBC 1997</td>
<td>1607.5, page</td>
<td>Equation 7-1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R = r(A-150) ) for FPS units</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R = r(A-13.94) ) for SI units</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R = r(A-150) ) for FPS units</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R = r(A-13.94) ) for SI units</td>
</tr>
<tr>
<td>IBC 2003</td>
<td>1607.9.2, page 277</td>
<td>Equation 16-22</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R = r(A-150) ) for FPS units</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R = r(A-13.94) ) for SI units</td>
</tr>
</tbody>
</table>

where

\[ A = \text{area of floor supported by the member} \]
\[ R = \text{reduction in percentage} \]
\[ r = \text{rate of reduction equal to 0.08 for floors} \]

Notes

a. Only the rules for live load on *Floors* have been implemented. The rules for live load on *Roofs* have not been implemented.

b. Since the medium of application of this method is the FLOOR LOAD or ONEWAY LOAD feature, and since STAAD performs load generation on beams only, the rules of the above-mentioned sections of the code for vertical members (columns) has not been implemented. The distributed load on those members found to satisfy the requirements explained in the code would have a lowered value after the reduction is applied.

c. Equation (7-2) of UBC 97, (16-3) of IBC 2000 and (16-23) of IBC 2003 have not been implemented.

d. In the IBC 2000 and 2003 codes, the first note says “A reduction shall not be permitted in Group A occupancies.” In STAAD, there is no direct method for conveying to the program that the occupancy type is Group A. So, it is the user’s responsibility to ensure that when he/she decides to utilize the live load reduction feature, the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

e. In the UBC 97 code, the last paragraph of section 1607.5 states that “The live load reduction shall not exceed 40 percent in garages for the storage of private pleasure cars having a capacity of not more than nine passengers per vehicle.” Again, there is no method to convey to STAAD that the structure is a garage for storing private pleasure cars. Hence, it is the user’s responsibility to ensure that the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

f. Because all the three codes follow the same rules for reduction, no provision is made available in the command syntax for specifying the code name according to which the reduction is to be done.

Related Links

- Technical Reference of STAAD Commands
- TR.32 Loading Specifications
- STAAD.Pro 2671
- User Manual
**Technical Reference of STAAD Commands**

TR.32 Loading Specifications

- **G.15.3 Area, One-way, and Floor Loads** (on page 2338)
- **M. To add a floor load or one-way load** (on page 831)
- **G.15.3 Area, One-way, and Floor Loads** (on page 2338)

**TR.32.4.3 Floor Load Specification**

Used to distribute a pressure load onto all beams that define a closed loop assuming a two way distribution of load.

The Floor Load specification be applied to groups and can also use live load reduction per IBC or UBC codes.

**General Format**

```
FLOOR LOAD (SAVE { LOAD })
YRANGE f1 f2 FLOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (PRINT {MEMBER | LOAD})
```

or

```
XRANGE f1 f2 FLOAD f3 (YRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (PRINT {MEMBER | LOAD})
```

or

```
ZRANGE f1 f2 FLOAD f3 (XRA f4 f5 YRA f6 f7) { GX | GY | GZ } (PRINT {MEMBER | LOAD})
```

or

```
_FloorGroupNameFLOAD f3 { GX | GY | GZ } (INCLINED)
```

Where:

- **f1 f2** Global coordinate values to specify Y, X, or Z range. The load will be calculated for all members lying in that global plane within the first specified global coordinate range.

- **f3** The value of the load (unit weight over square length unit). If the global direction is omitted, then this load acts parallel to the positive global Y if command begins with YRA and based on the area projected on a X-Z plane. Similarly, for commands beginning with XRA, the load acts parallel to the positive global X and based on the area projected on a Y-Z plane. Similarly, for commands beginning with ZRA, the load acts parallel to the positive global Z and based on the area projected on a X-Y plane.

- **f4 - f7** Global coordinate values to define the corner points of the area on which the specified floor load (**f3**) acts. If not specified, the floor load will be calculated for all members in all floors within the first specified global coordinate range.

- **GX, GY, GZ** If a Global direction is included, then the load is re-directed to act in the specified direction(s) with a magnitude of the loads which is based on the area projected on a plane as if the Global direction was omitted. The Global direction option is especially useful in mass definition.

- **FloorGroupName** See TR.16.1 Listing of Entities by Specifying Groups (on page 2440) for the procedure for creating FLOORGROUPs. The member-list contained in this name will be the candidates that will receive the load generated from the floor pressure.

- **INCLINED** - This option must be used when a FLOOR LOAD is applied on a set of members that form a panel(s) which is inclined to the global XY, YZ, or ZX planes.

- **PRINT** = This option is used when floor panel information printout is required. The total number of panels identified, total area of all panels and total load generated will be printed in the output file.
PRINT MEMBER = This option is used to print panel member numbers on which loads are generated along with the total number of panels identified, total area of all panels and total load generated.

PRINT LOAD = This option is used to print panel loading information. All other information available with PRINT MEMBER will also be printed in the output file.

SAVE = This option will give panel information in an external text file named \((filename)\)_FLD.TXT. The information corresponding to PRINT MEMBER will be printed in this file.

SAVE LOAD = This option will give panel information in an external text file named \((filename)\)_FLD.TXT. The information corresponding to PRINT LOAD will be printed in this file.

**Note:** It is recommended to use SAVE or SAVE LOAD option which will give same output printed in an external text file. If detailed output is printed in output (.ANL file) file using PRINT MEMBER or PRINT LOAD option, the size of the output file may increase to a large extent.

**Tip:** The SET FLOOR LOAD TOLERANCE command may be used to specify a specify the tolerance for out-of-plane nodes to be included in a floor load. See TR.5 Set Command Specification (on page 2413)

**Notes**

- **a.** The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).
- **b.** For the FLOOR LOAD specification, a two-way distribution of the load is considered. For the ONEWAY and AREA LOAD specification, a one-way action is considered. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the FLOOR LOAD type.
- **c.** FLOOR LOAD from a slab is distributed on the adjoining members as trapezoidal and triangular loads depending on the length of the sides as shown in the diagram. Internally, these loads are converted to multiple point loads.

![Diagram](image)

**Figure 312:** Members 1 and 2 get full trapezoidal and triangular loads respectively. Members 3 and 4 get partial trapezoidal loads and 5 and 6 get partial triangular load.

- **d.** The load per unit area may not vary for a particular panel and it is assumed to be continuous and without holes.
- **e.** The FLOOR LOAD facility is not available if the SET Z UP command is used (See Section 5.5.)
f. If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better.

If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better.

The attached example illustrates a case where the floor has to be sub-divided into smaller regions for the floor load generation to yield proper results. The internal angle at node 6 between the sides 108 and 111 exceeds 180 degrees. A similar situation exists at node 7 also. As a result, the following command:

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35
```

will not yield acceptable results. Instead, the region should be subdivided as shown in the following example

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35 XRA -.01 15.1 ZRA -0.1 8.1
YRANGE 11.9 12.1 FLOAD -0.35 XRA 4.9 10.1 ZRA 7.9 16.1
```

![Floor Plan](image)

g. The global horizontal direction options (GX and GZ) enables one to consider AREA LOADs, ONEWAY LOADs and FLOOR LOADs for mass matrix for frequency calculations.

h. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated on the members circumscribing those panels. In such cases, one ought to use the FLOOR LOAD type.

**Applying FLOOR LOAD onto a Floor Group**

When applying a floor load using XRANGE, YRANGE and ZRANGE, there are two limitations that one may encounter:

a. If panels consist of members whose longitudinal axis cross each other in an X type, and if the members are not connected to each other at the point of crossing, the panel identification and hence the load generation in that panel may fail. A typical such situation is shown in the plan drawing shown in the next figure.
After the load is specified, if the user decides to change the geometry of the structure (X, Y or Z coordinates of the nodes of the regions over which the floor load is applied), she/he has to go back to the load and modify its data too, such as the XRANGE, YRANGE and ZRANGE values. In other words, the 2 sets of data are not automatically linked.

The above limitations may be overcome using a FLOOR GROUP. A GROUP name is a facility which enables us to cluster a set of entities – nodes, members, plates, solids, etc. into a single moniker through which one can address them. Details of this are available in section 5.16 of this manual.

The syntax of this command, as explained earlier in this section is:

```
FLOOR LOAD

Floor-group-name FLOAD f3 { GX | GY | GZ }
```

Where:

```
f3 = pressure on the floor
```

To create equal loads in all 3 global directions for mass definition or other reasons, then enter direction labels for each direction desired; GY first then GX and/or GZ.

**Example**

```
START GROUP DEFINITION
FLOOR
  _PNL5A  21 22 23 28
END GROUP DEFINITION
LOAD 2 FLOOR LOAD on intermediate panel @ Y = 10 ft
FLOOR LOAD
  _PNL5A FLOAD -0.45 GY
  _PNL5A FLOAD -0.45 GY GX GZ

LOAD 5 LOAD ON SLOPING ROOF
FLOOR LOAD
  _SLOPINGROOF FLOAD -0.5 GY INCLINED
```
Live Load Reduction per UBC and IBC Codes

The UBC 1997, IBC 2000, and IBC 2003 codes permit reduction of floor live loads under certain situations. The provisions of these codes have been incorporated in the manner described further below.

To utilize this facility, the following conditions have to be met when creating the STAAD.Pro model.

i. The live load must be applied using the FLOOR LOAD or ONEWAY LOAD option. This option is described above, and an example of its usage may be found in EX. US-15 Wind and Floor Load Generation on a Space Frame (on page 4480).

ii. As shown in TR.32 Loading Specifications (on page 2650), the load case has to be assigned a Type called Live at the time of creation of that case. Additionally, the option called Reducible, also has to be specified as shown.

LOAD n LOADTYPE Live REDUCIBLE

Where:

n is the load case number

The following figures show the load generated on members for the two situations.

![Figure 314: Load generated for the previously described cases](image-url)
### Table 256: Table - Details of the code implementation

<table>
<thead>
<tr>
<th>Code Name</th>
<th>Section of code which has been implemented</th>
<th>Applicable Equations</th>
</tr>
</thead>
<tbody>
<tr>
<td>UBC 1997</td>
<td>1607.5, page</td>
<td>Equation 7-1&lt;br&gt;R = r(A-150) for FPS units&lt;br&gt;R = r(A-13.94) for SI units</td>
</tr>
<tr>
<td>IBC 2000</td>
<td>1607.9.2, page 302</td>
<td>Equation 16-2&lt;br&gt;R = r(A-150) for FPS units&lt;br&gt;R = r(A-13.94) for SI units</td>
</tr>
<tr>
<td>IBC 2003</td>
<td>1607.9.2, page 277</td>
<td>Equation 16-22&lt;br&gt;R = r(A-150) for FPS units&lt;br&gt;R = r(A-13.94) for SI units</td>
</tr>
</tbody>
</table>

where<br>\[ A = \text{area of floor supported by the member} \]<br>\[ R = \text{reduction in percentage} \]<br>\[ r = \text{rate of reduction equal to 0.08 for floors} \]

### Notes

**a.** Only the rules for live load on **Floors** have been implemented. The rules for live load on **Roofs** have not been implemented.

**b.** Since the medium of application of this method is the FLOOR LOAD or ONEWAY LOAD feature, and since STAAD performs load generation on beams only, the rules of the above-mentioned sections of the code for vertical members (columns) has not been implemented. The distributed load on those members found to satisfy the requirements explained in the code would have a lowered value after the reduction is applied.

**c.** Equation (7-2) of UBC 97, (16-3) of IBC 2000 and (16-23) of IBC 2003 have not been implemented.

**d.** In the IBC 2000 and 2003 codes, the first note says “A reduction shall not be permitted in Group A occupancies.” In STAAD, there is no direct method for conveying to the program that the occupancy type is Group A. So, it is the user’s responsibility to ensure that when he/she decides to utilize the live load reduction feature, the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

**e.** In the UBC 97 code, the last paragraph of section 1607.5 states that “The live load reduction shall not exceed 40 percent in garages for the storage of private pleasure cars having a capacity of not more than nine passengers per vehicle.” Again, there is no method to convey to STAAD that the structure is a garage for storing private pleasure cars. Hence, it is the user’s responsibility to ensure that the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

**f.** Because all the three codes follow the same rules for reduction, no provision is made available in the command syntax for specifying the code name according to which the reduction is to be done.

### Related Links

- Technical Reference of STAAD Commands
- TR.32 Loading Specifications
- STAAD.Pro 2677 User Manual
TR.32.5 Prestress Load Specification

This command may be used to specify PRESTRESS loads on members of the structure.

General Format

```
MEMBER {PRESTRESS | POSTSTRESS } (LOAD)
memb-list FORCE f1 *{ ES f2 | EM f3 | EE f4 }
```

Where:

- **FORCE f1** Prestressing force. A positive value indicates precompression in the direction of the local x-axis. A negative value indicates pretension.
- **ES f2** specifies eccentricity (from the centroid) of the prestress force at the start of the member.
- **EM f3** specifies eccentricity (from the centroid) of the prestress force at the mid-point of the member.
- **EE f4** specifies eccentricity (from the centroid) of the prestress force at the end of the member.

Description

The first option, (MEMBER PRESTRESS LOAD), considers the effect of the prestressing force during its application. Thus, transverse shear generated at the ends of the member(s) subject to the prestressing force is transferred to the adjacent members.

The second option, (MEMBER POSTSTRESS LOAD), considers the effect of the existing prestress load after the prestressing operation. Thus, transverse shear at the ends of the member(s) subject to the prestressing force is not transferred to the adjacent members.

Example

```
MEMBER PRESTRESS
  2 TO 7 11 FORCE 50.0
MEMBER POSTSTRESS
  8 FORCE 30.0 ES 3.0 EM -6.0 EE 3.0
```

In the first example, a prestressing force of 50 force units is applied through the centroid (i.e., no eccentricity) of members 2 to 7 and 11. In the second example, a poststressing force of 30 force units is applied with an eccentricity of 3 length units at the start, -6.0 at the middle, and 3.0 at the end of member 8.

One of the limitations in using this command is that under any one load case, on any given member, a prestress or poststress load may be applied only once. If the given member carries multiple stressed cables or has a PRESTRESS and POSTSTRESS load condition, such a situation will have to be specified through multiple load cases for that member. See example below.
Incorrect Input:
```plaintext
LOAD 1
MEMBER PRESTRESS
  6 7 FORCE 100 ES 2 EM -3 EE 2
  6 FORCE 150 ES 3 EM -6 EE 3
PERFORM ANALYSIS
```

Correct Input:
```plaintext
LOAD 1 MEMBER PRESTRESS
  6 7 FORCE 100 ES 2 EM -3 EE 2
LOAD 2
MEMBER PRESTRESS
  6 FORCE 150 ES 3 EM -6 EE 3
LOAD COMBINATION 3
  1 1.0 2 1.0
PERFORM ANALYSIS
```

Examples for Modeling Techniques

The following examples describe the partial input data for the members and cable profiles shown below.

![Figure 315: Example 1](image)

```
JOINT COORD
  1 0 0 ; 2 10 0
MEMBER INCI
  1 1 2
  ...
UNIT ...
MEMBER POSTSTRESS
  1 FORCE 100 ES 3 EM -3 EE 3
PERFORM ANALYSIS
```
**Figure 316: Example 2**

```
JOINT COORD
1 0 0 ; 2 20 0
MEMBER INCI
1 1 2
...
UNIT ...
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES -3 EM -3 EE -3
PERFORM ANALYSIS
```

**Figure 317: Example 3**

```
JOINT COORD
1 0 0 ; 2 5 0 ; 3 15 0 0 ; 4 20 0
MEMBER INCI
1 1 2 ; 2 2 3 ; 3 3 4
...
UNIT ...
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM -3 EE -3
3 FORCE 100 ES -3 EM 0 EE 3
PERFORM ANALYSIS
```
Figure 318: Example 4

JOINT COORD
1 0 0 ; 2 10 0 ; 3 20 0 0
MEMBER INCI
1 1 2 ; 2 2 3
...
UNIT ...
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM 0 EE 3
PERFORM ANALYSIS

Figure 319: Example 5

JOINT COORD
1 0 0 ; 2 10 0 ; 3 20 0 0
MEMBER INCI
1 1 2 ; 2 2 3
...
UNIT ...
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM -3 EE 3
2 FORCE 100 ES -3 EM -3 EE 3
PERFORM ANALYSIS

Related Links
- G.15.5 Prestress and Poststress Member Load (on page 2340)
- M. To add a prestress or post-tension load to members (on page 828)
- G.15.5 Prestress and Poststress Member Load (on page 2340)
TR.32.6 Temperature Load Specification for Members, Plates, and Solids

This command may be used to specify TEMPERATURE loads or strain loads on members, plates, and solids; or strain loads on members.

**General Format**

```
TEMPERATURE LOAD
memb/elem-list { TEMP f1 f2 f4 | STRAIN f3 | STRAINRATE f5 }
```

Where:

- **f1** The change in temperature which will cause axial elongation in the members or uniform volume expansion in plates and solids. The temperature unit is the same as the unit chosen for the coefficient of thermal expansion ALPHA under the CONSTANT command. (Members/Plates/Solids)

- **f2** The temperature differential from the top to the bottom of the member or plate ($T_{top} - T_{bottom}$). If $f2$ is omitted, no bending will be considered. (Local Y axis) (Members/Plates). Section depth must be entered for prismatic.

- **f3** Initial axial elongation (+)/shrinkage (-) in member due to misfit, etc. in length unit (Members only).

- **f4** The temperature differential from side to side of the member. (Local Z axis) (Members). Section or flange width must be entered for prismatic.

- **f5** Initial axial elongation (+)/shrinkage (-) per unit length, of members only.

**Example**

```
UNIT MMS
TEMP LOAD
  1 TO 9 15 17 TEMP 70.0
  18 TO 23 TEMP 90.0 66.0
  8 TO 13 STRAIN 3.0
  15 27 STRAINRATE 0.4E-4
```

**Notes**

**a.** It is not necessary or possible to specify the units for temperature or for ALPHA. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for the temperature load. (See TR.26 Specifying and Assigning Material Constants (on page 2501)).

If ALPHA was provided by a material name (STEEL, CONCRETE, ALUMINUM) then the temperature here must be in degree Fahrenheit units (if English length units were in effect when Alpha was defined) or degree Celsius units (if metric length units were in effect when Alpha was defined).

**Related Links**

- [G.15.6 Temperature and Strain Load](on page 2341)
TR.32.7 Fixed-End Load Specification

This command may be used to specify FIXED END loads on members (beams only) of the structure.

**General Format**

```plaintext
FIXED ( END ) LOAD
memb-list FXLOAD f1, f2, ... f12
```

Where:

- `memb-list`: normal STAAD.Pro member list rules (TO and BY for generation; and - to continue list to next line).
- `f1 ... f6`: Force-x, shear-y, shear-z, torsion, moment-y, moment-z (all in local coordinates) at the start of the member.
- `f7 ... f12`: Same as above except at the end of the member.

If less than 12 load values are entered, zero values will be added to the end. The loads may be extended to one additional line by ending the first load line with a - (hyphen).

These loads are given in the member local coordinate system and the directions are opposite to the actual load on the member.

**Related Links**

- [G.15.4 Fixed End Member Load](on page 2340)
- [M. To add fixed end member loads](on page 829)
- [G.15.4 Fixed End Member Load](on page 2340)

TR.32.8 Support Joint Displacement Specification

This command may be used to specify displacements (or generate loads to induce specified displacements) in supported directions (pinned, fixed, enforced, or spring).

**General Format**

```plaintext
SUPPORT DISPLACEMENT
support-joint-list { FX | FY | FZ | MX | MY | MZ } f1
```

Where:

- `f1`: Value of the corresponding displacement. For translational displacements, the unit is in the currently specified length unit, while for rotational displacements the unit is always in degrees.

FX, FY, FZ specify translational displacements in X, Y, and Z directions respectively. MX, MY, MZ specify rotational displacements in X, Y, and Z directions.

**Description**

There are two distinct modes of usage for this command. If any ENFORCED specifications were used in the support command then the “displacement mode” is used; otherwise the “load mode” is used. Despite the name
of this command, if displacements are specified in spring directions, the displacement is at the joint not at the 
grounded end of the support spring. Displacement cannot be specified in a direction that does not have a 
support or a spring.

1. DISPLACEMENT MODE - With this mode, the support joint displacement is modeled as an imposed joint 
displacement. The joint directions where displacement may be specified must be defined (same for all cases) 
in the SUPPORT command, see TR.27.1 Global Support Specification (on page 2514). Any beam members, 
springs or finite elements will be considered in the analysis. Other loading, inclined supports, and master/ 
slave are all considered. Any number of cases may have displacements entered. However, all cases will have 
zero displacements at the enforced directions if no displacement values are entered for that case. At inclined 
supports the displacement specification is assumed to be in the inclined direction. Displacements may not be 
specified at slave directions.

If some cases are to have spring supports and others enforced displacements at the same joint directions, 
then two PERFORM ANALYSES must be used with the CHANGE command in between. The first perform 
analysis could have the SUPPORTS with springs, no enforced directions, and with the load cases without 
displacements. The second perform analysis would then have SUPPORTS without springs but with enforced 
directions and the cases with displacements.

Displacement Mode Restrictions:
• The Support Displacement command may be entered only once per case.
• Spring directions and Enforced directions may not both be specified at the same joint direction in the 
same Perform Analysis step.

2. LOAD MODE - With this mode, the support joint displacement is modeled as a load. Only beam members 
(not springs or finite elements) are considered in computing the joint load distribution necessary to cause 
the displacement. Other loading, inclined supports, and master/slave are also not considered. These 
unconsidered factors, if entered, will result in displacements other than those entered (results are 
superimposed). Only those cases with displacements entered will be affected.

Load Mode Restrictions:
• Support Displacements can be applied in up to four load cases only.
• The Support Displacement command may be entered only once per case.
• Finite elements should not be entered.
• Inclined supports must not be entered.
• Spring supports are not considered in calculating the load so their use will lead to displacements different 
from the input values.

Example
UNIT ...
SUPPORT DISPL
  5 TO 11 13 FY -0.25
  19 21 TO 25 MX 15.0

In this example, the joints of the first support list will be displaced by 0.25 units in the negative global 
Y direction. The joints of the second support list will be rotated by 15 degrees about the global X-axis.

Related Links
• G.15.7 Support Displacement Loads (on page 2342)
• M. To add a support displacement (on page 825)
TR.32.9 Selfweight

Selfweight commands are used to calculate and apply the self weight of structural elements for analysis.

TR.32.9.1 Selfweight Loads

Used to calculate and apply the self weight of members, plates, and solids in the structure for analysis.

General Format

```
SELFWEIGHT ( { X | Y | Z } f1 ) ( LIST member-list | ALL )
```

Where:

- **X, Y, & Z** represent the global direction in which the selfweight acts. The default direction is along the Y axis.
- **f1** The factor to be used to multiply the selfweight. The default value is -1.0.
- **member-list** member, plate and/or solid list or group. All list specifications such as explicit list including TO and BY, range (XR, YR,ZR), parallel to (X, Y, Z) and groups are supported. If a group is used, the selfweight command must be repeated for each group. If list is not provided, all structural components are used for body weight calculation unless those are inactive either by INACTIVE command specification or internally set due to TENSION/COMPRESSION ONLY specification.

Description

This command is used if the self-weight of the members, plates, and solids within the structure is to be considered. The self-weight of every active element is calculated and applied as a uniformly distributed member load.

**Note:** Surfaces are not included in the selfweight command. The SSELFwT must be used to include the selfweight for surface elements. See TR.32.9.2 Surface Selfweight Load (on page 2686)

This command may also be used without any direction and factor specification. Thus, if specified as SELFWEIGHT, loads will be applied in the negative global Y direction with a factor of unity.

Notes

- **a.** Density must be provided for calculation of the self weight.
- **b.** The selfweight of finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.
- **c.** The selfweight of a plate is placed at the joints, regardless of plate releases.
- **d.** The SELFWEIGHT specification for definition of static method of seismic load generator also is capable of accepting a list. Only there is no direction specification for this command.
- **e.** Similarly, the SELFWEIGHT specification when used in a Reference Load is also capable of handling a list.

Example

```
LOAD 1 DEAD AND LIVE LOAD
* includes the selfweight of all members, plates, and solids. The following
```
two commands are functionally identical
SELF
SELFWEIGHT y -1.0 all
*
* Factored weight for members, plates, and solids 4 through 10 along global x
direction
SELFWEIGHT X 1.4 LIST 4 TO 10
* Includes weight of all members, plates, and solids associated with the group
_PLATEGRP1
SELF X 1.0 _PLATEGRP1

Related Links

• M. To add selfweight load (on page 823)

TR.32.9.2 Surface Selfweight Load

Used to calculate and apply the self weight of surface elements in the structure for analysis.

**Note:** This feature requires STAAD.Pro V8i (SELECTseries 3) or higher.

General Format

\[
\text{SSELFWEIGHT} \left( \{ X \mid Y \mid Z \} \ f1 \right) \ (\text{LIST} \ \text{surface-list} \mid \text{ALL})
\]

Where:

- **X, Y, & Z** represent the global direction in which the selfweight acts. The default direction is along the Y axis.
- **f1** The factor to be used to multiply the selfweight. The default value is -1.0.
- **surface-list** surface element list or group. All list specifications such as explicit list including TO and BY, range (XR, YR, ZR), parallel to (X, Y, Z) and groups are supported. If a group is used, the selfweight command must be repeated for each group. If list is not provided, all structural components are used for body weight calculation unless those are inactive either by INACTIVE command specification or internally set due to TENSION/COMPRESSION ONLY specification.

See TR.32.1 Selfweight Loads (on page 2685) for Description and Notes pertaining to self weight.

**Example**

```
LOAD 1 LOADTYPE none TITLE DEAD LOAD
SURFACE LOAD
  1 PR 1 0 0
SSELFWEIGHT Y -1
```

Related Links

• M. To add a surface selfweight load (on page 832)

TR.32.10 Dynamic Loading Specification

The command specification needed to perform response spectrum analysis and time-history analysis is explained in the following sections.
Note: STAAD.Pro is also capable of generating floor spectrum responses for a time history analysis. Refer to TR.37.10 Floor Spectrum Command (on page 2833) for details on adding this to the analysis commands.

Related Links
- G.17.3.1 Solution of the Eigenproblem (on page 2363)
- G.15.1 Joint Loads (on page 2337)
- G.17.3 Dynamic Analysis (on page 2362)

TR.32.10.1 Response Spectrum Analysis

Various methods for performing response spectrum analysis have been implemented in STAAD.Pro. They include a generic method that is described in most textbooks, as well as code-based methods like those required by the IBC, Eurocode 8, IS 1893, etc. These are described in the following sections.

Table 257: Codes available in STAAD.Pro with Response Spectrum loads

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Canada</td>
<td>NRC 2005 (on page 2694)</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td></td>
<td>NRC 2010 (on page 2702)</td>
<td>National Building Code(NRC/CNRC) of Canada</td>
</tr>
<tr>
<td>Russia</td>
<td>SNiP II-7-81 (on page 2759)</td>
<td>Строительство в сейсмических районах (Construction in Seismic Regions)</td>
</tr>
</tbody>
</table>
### Technical Reference of STAAD Commands

#### TR.32 Loading Specifications

<table>
<thead>
<tr>
<th>Country</th>
<th>Code</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><strong>SP 14.13330.2011</strong> (on page 2764)</td>
<td>Строительство в сейсмических районах (Construction in Seismic Regions)</td>
</tr>
</tbody>
</table>

#### Related Links

- **G.17.3.4 Response Spectrum** (on page 2371)

### TR.32.10.1.1 Response Spectrum Specification - Custom

This command may be used to specify and apply a custom (i.e., “generic” method) RESPONSE SPECTRUM loading for dynamic analysis.

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. The maximum number of response spectrum load cases allowed in one run is 50.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, refer to **TR.32 Loading Specifications** (on page 2650) and **G.17.3 Dynamic Analysis** (on page 2362). An illustration of mass modeling is available, with explanatory comments, in Example Problem No.11.

#### General Format

```
SPECTRUM comb-method *{ X f1 | Y f2 | Z f3 } { ACCELERATION | DISPLACEMENT } (SCALE f4)
{DAMP f5 | CDAMP | MDAMP } ( { LINEAR | LOGARITHMIC } ) (MISSING f6) (ZPA f7) (FF1 f8)
(FF2 f9) ( { DOMINANT f10 | SIGN } ) (SAVE) (IMR f11) (STARTCASE f12)
```

**Note:** The data from SPECTRUM through SCALE above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, enter Spectra in one of these two input forms (i.e., explicit values or an external file):
```
{ p1 v1; p2 v2; p3 v3; ... | FILE filename }
```

Where:
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X $f_1$, Y $f_2$, Z $f_3$</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>SCALE $f_4$</td>
<td>1.0</td>
<td>Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec$^2$ units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s$^2$ or 32.2 ft/s$^2$).</td>
</tr>
<tr>
<td>DAMP $f_5$</td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.</td>
</tr>
<tr>
<td>MISSING $f_6$</td>
<td>0</td>
<td>Optional parameter to use the “Missing Mass” method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec$^2$ for this missing mass mode is the $f_6$ value entered in length per second squared units (this value is not multiplied by SCALE). If $f_6$ is zero, then the spectral acceleration at the ZPA $f_7$ frequency is used. If $f_7$ is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results.</td>
</tr>
<tr>
<td>ZPA $f_7$</td>
<td>33 [Hz]</td>
<td>The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING $f_6$ is entered.</td>
</tr>
<tr>
<td>FF1 $f_8$</td>
<td>2 [Hz]</td>
<td>The $f_1$ parameter defined in the ASCE 4-98 standard in Hz units. For ASCE option only.</td>
</tr>
</tbody>
</table>
### Parameter Defaults

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FF2 $f_9$</td>
<td>33 [Hz]</td>
<td>The $f_2$ parameter defined in the ASCE 4-98 standard in Hz units. For ASCE option only.</td>
</tr>
<tr>
<td>DOMINANT $f_{10}$</td>
<td>1 (1st Mode)</td>
<td>The dominant mode method. All results will have the same sign as mode number $f_{10}$ alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). <strong>Note:</strong> Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</td>
</tr>
<tr>
<td>IMR $f_{11}$</td>
<td>1</td>
<td>The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through $f_{11}$ will be used. Missing Mass modes are not output.</td>
</tr>
<tr>
<td>STARTCASE $f_{12}$</td>
<td>Highest Load Case No. + 1</td>
<td>The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If $f_{12}$ is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.</td>
</tr>
</tbody>
</table>

### Combining Methods

$comb\text{-}method = \{\text{ SRSS } | \text{ ABS } | \text{ CQC } | \text{ ASCE } | \text{ TEN } | \text{ CSM } | \text{ GRP } \}$ are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

- **SRSS** Square Root of Summation of Squares method.
- **ABS** Absolute sum. This method is very conservative and represents a worst case combination.
- **CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[
F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}
\]

where

\[
\rho_{nm} = \frac{8\xi^2(1 + r)r^{2/3}}{(1 - r^2)^{2} + 4\xi^2r(1 + r)^2}
\]

\[
r = \frac{\omega_n/\omega_m - 1}{2}
\]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.
ASCE  NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


CSM  Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.


Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

ACCELERATION or DISPLACEMENT  indicates whether Acceleration or Displacement spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

\[
\text{Displacement} = \text{Acceleration} \times \left(\frac{1}{\omega}\right)^2
\]

where

\[
\omega = \frac{2\pi}{\text{Period}} \quad \text{(period given in seconds; \(\omega\) in cycles per second)}
\]

DAMP, MDAMP, and CDAMP  select source of damping input:

- DAMP indicates to use the \(f_2\) value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode.

You must specify damping for different materials under the CONSTANT specification.

LINEAR or LOGARITHMIC  Select Linear or Logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When FILE filename is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

SAVE  This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g’s and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

SIGN  This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

Caution: Do not enter DOMINANT parameter with this option.

Spectra data is input in one of these two input forms:
1. \( p_1, v_1; p_2, v_2; \ldots; p_n, v_n \). Data is part of input, immediately following the SPECTRUM command. Period – Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Period is in seconds and the corresponding Value is either acceleration (current length unit/sec\(^2\)) or displacement (current length unit) depending on the ACC or DIS chosen. Continue the curve data onto as many lines as needed (up to 500 spectrum pairs). Spectrum pairs must be in ascending order of period. Note, if data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.81, 386.4, etc.). Also note, do not end these lines with a hyphen (-). Each SPECTRUM command must be followed by Spectra data if this input form is used.

2. FILE \textit{filename} data is in a separate file, using the format described in File Format for Spectra Data (on page 2693).

When the FILE \textit{filename} command has been provided, then you must have the spectra curve data on a file named \textit{filename} prior to starting the analysis. The format of the FILE spectra data allows spectra as a function of damping as well as period.

**Note:** If the FILE \textit{filename} command is entered, it must be with the first spectrum case and will be used for all spectrum cases.

No File \textit{filename} command needs to be entered with the remaining spectrum cases. The \textit{filename} may not be more than 72 characters in length.

### Examples

An example using joint loads and the SRSS combination method:

```
LOAD 2 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2
 0.20  0.2 ;  0.40  0.25 ;  0.60  0.35 ;  0.80  0.43 ;  1.0  0.47
 1.2  0.5 ;  1.4  0.65 ;  1.6  0.67 ;  1.8  0.55 ;  2.0  0.43
```

An example using member loads and the CQC combination method:

```
LOAD 2 SEISMIC LOADING
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
MEMBER LOADS
  5  CON  GX  5.0  6.0
  5  CON  GY  5.0  6.0
  5  CON  GX  7.5  10.0
  5  CON  GY  7.5  10.0
  5  CON  GX  5.0  14.0
  5  CON  GY  5.0  14.0
SPECTRUM CQC X 1.0 ACC DAMP 0.05 SCALE 32.2
 0.03  1.00 ;  0.05  1.35
 0.1  1.95 ;  0.2  2.80
 0.5  2.80 ;  1.0  1.60
```
Multiple Response Spectra

If there is more than one response spectrum defined in the input file, the load data (representing the dynamic weight) should accompany the first set of spectrum data only. In the subsequent load cases, only the spectra should be defined. See example below.

LOAD 1 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2 IMR 2 STARTCASE 11
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
PERFORM ANALYSIS
CHANGE

LOAD 2 SPECTRUM IN Y-DIRECTION
SPECTRUM SRSS Y 1.0 ACC SCALE 32.2
0.20 0.1 ; 0.40 0.15 ; 0.60 0.33 ; 0.80 0.45 ; 1.00 0.48
1.20 0.51 ; 1.4 0.63 ; 1.6 0.67 ; 1.8 0.54 ; 2.0 0.42

File Format for Spectra Data

The format of the FILE spectra data allows spectra as a function of damping as well as period. The format is:

<table>
<thead>
<tr>
<th>Dataset 1</th>
<th>MDAMPCV NPOINTCV</th>
</tr>
</thead>
<tbody>
<tr>
<td>(no of values = 2)</td>
<td></td>
</tr>
<tr>
<td>Dataset 2</td>
<td>Damping Values in ascending order</td>
</tr>
<tr>
<td>(no of values = Mdampcv)</td>
<td></td>
</tr>
<tr>
<td>Dataset 3a</td>
<td>Periods</td>
</tr>
<tr>
<td>(no of values = Npointcv)</td>
<td></td>
</tr>
<tr>
<td>Dataset 3b</td>
<td>Spectra</td>
</tr>
<tr>
<td>(no of values = Npointcv)</td>
<td></td>
</tr>
</tbody>
</table>

For ASCE, the MIS option is assumed to be on. If any of f6, f7, f8, f9 are not entered the defaults will be used.

Repeat Data set 3 $Mdampcv$ times (3a,3b,3a,3b,3a,3b,etc.) (i.e., for each damping value).

Data sets 2, 3a and 3b must have exactly $Npointcv$ values each. Blanks or commas separate the values. The data may extend to several lines. Do not end lines with a hyphen (-). No comment lines (*) or semi-colons. Multiple values may be entered per line.

where

<table>
<thead>
<tr>
<th>MDAMPCV</th>
<th>= Number of damping values for which there will be separate Spectra vs. Period curves.</th>
</tr>
</thead>
<tbody>
<tr>
<td>NPOINTCV</td>
<td>= Number of points in each Spectra vs. Period curve. If NPOINTCV is negative, then the period-spectra values are entered as pairs.</td>
</tr>
</tbody>
</table>

Examples of Spectra Data files

An example of spectral data for use in the X direction:

<table>
<thead>
<tr>
<th>1.0</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05</td>
<td></td>
</tr>
<tr>
<td>0.20 0.2 0.40 0.25 0.60 0.35 0.80 0.43 1.0 0.47</td>
<td></td>
</tr>
<tr>
<td>1.2 0.5 1.4 0.65 1.6 0.67 1.8 0.55 2.0 0.43</td>
<td></td>
</tr>
</tbody>
</table>
Individual Modal Response Case Generation

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.

Related Links

- M. To add a generic response spectrum (on page 843)
- G.17.3.4 Response Spectrum (on page 2371)
- M. To add an IBC 2000 response spectrum (on page 844)
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3 Dynamic Analysis (on page 2362)

TR.32.10.1.2 Response Spectrum Specification per NRC 2005

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2005 edition of the National Research Council specification National Building Code of Canada (NBC), for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.
General Format

SPECTRUM _comb-method_ NRC 2005 (TORSION) (DECCENTRICITY _f20_) (ECCENTRICITY _f21_) *{ _X f1_ _Y f2_ _Z f3_} { ACC | DIS } ( SCALE _f4_) 

{ DAMP _f5_ | CDAMP | MDAMP } ( { LINEAR | LOGARITHMIC } ) (MISSING _f6_) (ZPA _f7_)

{{ DOMINANT _f10_ | SIGN }} (SAVE) (IMR _f11_) (STARTCASE _f12_)

Note: The data from SPECTRUM through SCALE must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following spectrum data which must be started on a new line:

{ _p1 v1_; _p2 v2_; _p3 v3_; … _pn vn_ | FILE filename }

Where:

Table 259: Parameters used for NRC 2005 response spectrum

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DECCENTRICITY <em>f20</em></td>
<td>1.0</td>
<td>Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).</td>
</tr>
<tr>
<td>ECCENTRICITY <em>f21</em></td>
<td>0.05</td>
<td>Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.</td>
</tr>
<tr>
<td><em>X f1, Y f2, Z f3</em></td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>SCALE <em>f4</em></td>
<td>1.0</td>
<td>Linear scale factor by which the spectra data will be multiplied. Usually used to factor g’s to length/sec² units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s² or 32.2 ft/s²).</td>
</tr>
<tr>
<td>DAMP <em>f5</em></td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.</td>
</tr>
</tbody>
</table>

If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)

If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510)
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
</table>
| MISSING $f_6$   | Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the $f_6$ value entered in length/sec$^2$ (this value is not multiplied by SCALE).  
If $f_6$ is zero, then the spectral acceleration at the ZPA $f_7$ frequency is used. If $f_7$ is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results.  
**Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.  
**Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode. |                                                                                                                                                                                                                                                                                                                                 |
| ZPA $f_7$       | 33 [Hz]       | The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING $f_6$ is entered.                                                                                                                                                                                                                       |
| DOMINANT $f_{10}$ | 1 (1st Mode) | The dominant mode method. All results will have the same sign as mode number $f_{10}$ alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).  
**Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode. |                                                                                                                                                                                                                                                                                                                                 |
| IMR $f_{11}$    | 1             | The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through $f_{11}$ will be used. Missing Mass modes are not output.                                                                                                                   |
| STARTCASE $f_{12}$ | Highest Load Case No. + 1 | The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If $f_{12}$ is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one. |                                                                                                                                                                                                                                                                                                                                 |

$comb-method = \{ \text{SRSS} \mid \text{ABS} \mid \text{CQC} \mid \text{ASCE} \mid \text{TEN} \mid \text{CSM} \mid \text{GRP} \}$ are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

**SRSS**  Square Root of Summation of Squares method.

**ABS**  Absolute sum. This method is very conservative and represents a worst case combination.
CQC  Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \sqrt{\sum_n \sum_m f_n^2 \rho_{nm} f_m^2} \]

where

\[ \rho_{nm} = \frac{8 \zeta^2 (1 + r) r^{2/3}}{(1 - r^2)^2 + 4 \zeta^2 r (1 + r)^2} \]

\[ r = \frac{\omega_n}{\omega_m} \leq 1.0 \]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.

ASCE  NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


CSM  Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.


**Note:** If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

TORSION  indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Inherent and Accidental Torsion (on page 2699) for additional information.

**Note:** If TORSION is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

ACCELERATION or DISPLACEMENT  indicates whether Acceleration or Displacement spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

\[ \text{Displacement} = \text{Acceleration} \times (1/\omega)^2 \]

where

\[ \omega = \frac{2\pi}{\text{Period}} \] (period given in seconds; \( \omega \) in cycles per second)

DAMP, MDAMP, and CDAMP  select source of damping input:

- DAMP indicates to use the \( f_2 \) value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification
LINEAR or LOGARITHMIC

Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When **FILE** `filename` is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

SIGN

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for **ABS** option.

**Caution:** Do **not** enter **DOMINANT** parameter with this option.

SAVE

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g’s and radians/sec$^2$. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Spectra data is input in one of these two input forms:

1. **p1, v1; p2, v2; ... ; pn, vn.** Data is part of input, immediately following the SPECTRUM command. Period – Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Period is in seconds and the corresponding Value is either acceleration (current length unit/sec$^2$) or displacement (current length unit) depending on the ACC or DIS chosen. Continue the curve data onto as many lines as needed (up to 500 spectrum pairs). Spectrum pairs must be in ascending order of period. Note, if data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.81, 386.4, etc.). Also, note, do not end these lines with a hyphen (-). Each SPECTRUM command must be followed by Spectra data if this input form is used.

2. **FILE `filename`** data is in a separate file, using the format described in [File Format for Spectra Data](on page 2693).

When the **File** `filename` command has been provided, then you must have the spectra curve data on a file named `filename` prior to starting the analysis. The format of the FILE spectra data allows spectra as a function of damping as well as period.

**Note:** If the **FILE** `filename` command is entered, it must be with the first spectrum case and will be used for all spectrum cases.

No **File** `filename` command needs to be entered with the remaining spectrum cases. The `filename` may not be more than 72 characters in length.

**Individual Modal Response Case Generation**

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.
The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

**Note:** When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.

TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688) for additional details on IMR load case generation.

**Inherent and Accidental Torsion**

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc.) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

**Steps**

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.

   Thus, design eccentricity $e_{di} = f2\theta e_{si} + f12b1$ where $f2\theta = 1.0$ and $f21 = (\pm) 0.05$ where

   $e_{di}$
3. Torsional moment is calculated at each floor.

\[ M_{ik} = Q_{ik} \times e_{di} \text{ at floor } i \text{ for mode } k \]

4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.

5. Static analysis of the structure is performed with these nodal forces.

6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

**Dynamic Eccentricity**

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

In most country's seismic codes, the provision for design eccentricity \( e_{di} \) at \( i^{th} \) floor of a building is given by the following equations:

\[
\begin{align*}
    e_{di} &= \alpha e_{si} + \beta b_{i} \\
    e_{di} &= \delta e_{si} - \beta b_{i}
\end{align*}
\]

where

- \( e_{si} \) = static eccentricity at \( i^{th} \) floor
- \( b_{i} \) = plan dimension of the \( i^{th} \) floor normal to the direction of ground motion
- \( \alpha, \beta, \delta \) = specified constants

If \( \alpha = 1.0, \delta = 1.0, \) and \( \beta = 0.05 \) parameters to be defined are \( \text{DEC} = 1.0 \) and \( \text{ACC} = 0.05 \) in one load case and \( \text{DEC} = 1.0 \) and \( \text{ACC} = -0.05 \) in another load case to include both natural and accidental torsion.

Only TOR ECC 0.05 or TOR ECC -0.05 can also be defined without specifying DEC 1.0 since it is the default that is included in the analysis.
The natural torsion is automatically included in analysis for $DEC \leq 1.0$ i.e., no additional inherent torsion is applied. However, if $DEC > 1.0$, a twisting moment with modified eccentricity of $DEC-1$ will act at CM. In this case, a message occurs in the output.

Example

```
LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS NRC 2005 X 0.5 ACC DAMP 0.05 LIN
0 9.80665; 0.06 18.6326; 0.12 24.5166; 0.18 24.5166; 0.24 24.5166; 0.3 24.5166;
0.36 24.5166; 0.42 23.3492; 0.48 20.4305; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6203; 0.78 12.5726; 0.84 11.6746; 0.8 10.8963;
0.9 10.2153; 0.96 9.61436; 1.02 9.08023; 1.08 8.60233; 1.14 8.17221;
1.2 7.8806; 1.26 7.42928; 1.32 7.0627; 1.38 6.71018; 1.44 6.3777;
1.5 6.2863; 1.56 6.05349; 1.6 5.56791; 1.66 5.3601; 1.72 5.14814;
1.78 4.9367; 1.84 4.73602; 1.9 4.5874; 1.96 4.45166; 2.02 4.3361;
2.08 4.24166; 2.14 4.16316; 2.2 4.09103; 2.26 3.98471; 2.32 3.85179;
2.38 3.7684; 2.44 3.6326; 2.5 3.47754; 2.56 3.3459; 2.62 3.1805;
2.68 3.0924; 2.74 3.0724; 2.8 3.0624; 2.86 3.0024; 2.92 2.9724;
2.98 2.9524; 3.04 2.9424; 3.1 2.9324; 3.16 2.9224; 3.22 2.9124;
3.28 2.9024; 3.34 2.8924; 3.4 2.8824; 3.46 2.8724; 3.52 2.8624;
3.58 2.8524; 3.64 2.8424; 3.7 2.8324; 3.76 2.8224; 3.82 2.8124;
3.88 2.8024; 3.94 2.7924; 4 2.7824; 4.06 2.7724; 4.12 2.7624;
4.18 2.7524; 4.24 2.7424; 4.3 2.7324; 4.36 2.7224; 4.42 2.7124;
4.48 2.7024; 4.54 2.6924; 4.6 2.6824; 4.66 2.6724; 4.72 2.6624;
4.78 2.6524; 4.84 2.6424; 4.9 2.6324; 4.96 2.6224; 5 2.6124;
```

Related Links
- V. NRC 2005 Response Spectrum (on page 3553)

**TR.32.10.1.3 Response Spectrum Specification per NRC 2010**

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2010 edition of the National Research Council specification National Building Code of Canada (NBC), for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

**General Format**

```
SPECTRUM comb-method NRC 2010 (TORSION) (DECCENTRICITY f20) (ECCENTRICITY f21) *{ X f1 | Y f2 | Z f3} { ACCELERATION | DISPLACEMENT } ( SCALE f4 )
{ DAMP f5 | CDAMP | MDAMP } ( {LINEAR | LOGARITHMIC } ) (MISSSSING f6) (ZPA f7) ( { DOMINANT f10 | SIGN }) (SAVE) (IMR f11) (STARTCASE f12)
```

**Note:** The data from SPECTRUM through SCALE must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following spectrum data which must be started on a new line:

```
{ p1 v1; p2 v2; p3 v3; ... pn vn | FILE filename }
```

**Table 260: Parameters used for NRC 2010 response spectrum**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DECCENTRICITY  f20</td>
<td>1.0</td>
<td>Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).</td>
</tr>
<tr>
<td>ECCENTRICITY  f21</td>
<td>0.05</td>
<td>Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.</td>
</tr>
<tr>
<td>X f1, Y f2, Z f3</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>SCALE f4</td>
<td>1.0</td>
<td>Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/ sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/ s^2 or 32.2 ft/ s^2).</td>
</tr>
</tbody>
</table>
### Technical Reference of STAAD Commands

**TR.32 Loading Specifications**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DAMP f5</strong></td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503). If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510).</td>
</tr>
<tr>
<td><strong>MISSING f6</strong></td>
<td></td>
<td>Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the f6 value entered in length/sec² (this value is not multiplied by SCALE). If f6 is zero, then the spectral acceleration at the ZPA f7 frequency is used. If f7 is zero or not entered, the spectral acceleration at 33 Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. <strong>Note:</strong> If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.</td>
</tr>
<tr>
<td><strong>ZPA f7</strong></td>
<td>33 [Hz]</td>
<td>The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING f6 is entered.</td>
</tr>
<tr>
<td><strong>DOMINANT f10</strong></td>
<td>1 (1st Mode)</td>
<td>The dominant mode method. All results will have the same sign as mode number f10 alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). <strong>Note:</strong> Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</td>
</tr>
<tr>
<td><strong>IMR f11</strong></td>
<td>1</td>
<td>The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.</td>
</tr>
<tr>
<td><strong>STARTCASE f12</strong></td>
<td>Highest Load Case No. + 1</td>
<td>The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If f12 is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.</td>
</tr>
</tbody>
</table>
comb-method = \{ \text{SRSS} | \text{ABS} | \text{CQC} | \text{ASCE} | \text{TEN} | \text{CSM} | \text{GRP} \} \text{ are methods of combining the responses from each mode into a total response.}

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

**SRSS** Square Root of Summation of Squares method.

**ABS** Absolute sum. This method is very conservative and represents a worst case combination.

**CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \sqrt{\sum_n \sum_m f_n f_m \rho_{nm}} \]

where

\[ \rho_{nm} = \frac{8 \zeta^2 (1 + r) r^2 / 3}{(1 - r^2)^2 + 4 \zeta^2 r (1 + r)^2} \]

\[ r = \frac{\omega_n}{\omega_m} \leq 1.0 \]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.

**ASCE** NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


**CSM** Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

**GRP** Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

**Note:** If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

**TORSION** indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Inherent and Accidental Torsion (on page 2699) for additional information.

**Note:** If TORSION is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

**ACCELERATION or DISPLACEMENT** indicates whether Acceleration or Displacement spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

\[ \text{Displacement} = \text{Acceleration} \times \left( \frac{1}{\omega} \right)^2 \]
where \[ \omega = \frac{2\pi}{\text{Period}} \] (period given in seconds; \( \omega \) in cycles per second)

**DAMP, MDAMP, and CDAMP**

Select source of damping input:

- DAMP indicates to use the \( f_2 \) value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode.

Note: You must specify damping for different materials under the CONSTANT specification.

**LINEAR or LOGARITHMIC**

Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When FILE _filename_ is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**SIGN**

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

**Caution:** Do not enter DOMINANT parameter with this option.

**SAVE**

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g’s and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

**Individual Modal Response Case Generation**

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/ floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.
**Note:** When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.

**TR.32.10.1.1 Response Spectrum Specification - Custom** (on page 2688) for additional details on IMR load case generation.

**Inherent and Accidental Torsion**

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc.) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

**Steps**

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.
   
   Thus, design eccentricity \( e_{di} = f2\theta \times e_{si} + f12 \times b_i \) where \( f2\theta = 1.0 \) and \( f21 = (\pm) 0.05 \)
   
   where
   
   \( e_{si} \) = static eccentricity between center of mass and center of rigidity at floor \( i \)
   
   \( b_i \) = floor plan dimension in the direction of earthquake loading
3. Torsional moment is calculated at each floor.
   
   \( M_{ik} = Q_{ik} \times e_{di} \) at floor \( i \) for mode \( k \)
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.
7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

**Dynamic Eccentricity**

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

In most country’s seismic codes, the provision for design eccentricity $e_{di}$ at $i$th floor of a building is given by the following equations:

$$
\begin{align*}
    e_{di} &= \alpha e_{si} + \beta b_i \\
    e_{di} &= \delta e_{si} - \beta b_i
\end{align*}
$$

where

- $e_{si}$ = static eccentricity at $i$th floor
- $b_i$ = plan dimension of the $i$th floor normal to the direction of ground motion
- $\alpha$, $\beta$, and $\delta$ = specified constants

If $\alpha = 1.0$, $\delta = 1.0$, and $\beta = 0.05$ parameters to be defined are DEC = 1.0 and ACC = 0.05 in one load case and DEC = 1.0 and ACC = -0.05 in another load case to include both natural and accidental torsion.

Only TOR ECC 0.05 or TOR ECC -0.05 can also be defined without specifying DEC 1.0 since it is the default that is included in the analysis.

- (a) Design case,
- (b) Pure translation case,
- (c) Twisting moment including additional torsional moment plus accidental torsion,
- (d) Load applied through CM,
The cases (d) and (e) are followed i.e. seismic load is passed through CM instead of CR. Natural torsion is automatically included in analysis for DEC ≤ 1.0 i.e., no additional inherent torsion is applied. However, if DEC > 1.0, a twisting moment with modified eccentricity of DEC - 1 will act at CM. In this case, a message occurs in the output.

Example

This example input file demonstrates a seismic load using the equivalent force method and a seismic response spectrum analysis per NRC 2010.
MEMBER PROPERTY CANADIAN
101 TO 136 201 TO 236 PRIS YD 0.4 ZD 0.3
301 TO 303 305 TO 307 319 TO 317 TO 319 321 TO 323 325 TO 327 333 TO 334 TO 335 337 TO 339 341 TO 343 345 TO 347 TABLE ST W460X52
304 308 312 TO 316 320 324 328 TO 332 336 340 344 348 TABLE ST W530X85
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
  E 2.5e+007
  POISSON 0.17
  DENSITY 24
  ISOTROPIC STEEL
  E 2.05e+008
  POISSON 0.3
  DENSITY 76.8195
  ALPHA 1.2e-005
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
  E 2.17185e+007
  POISSON 0.17
  DENSITY 23.5616
  ALPHA 1e-005
  DAMP 0.05
  TYPE CONCRETE
  STRENGTH FCU 27579
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 MEMB 101 TO 136 201 TO 236
MATERIAL STEEL MEMB 301 TO 348
SUPPORTS
1 TO 16 FIXED
CUT OFF MODE SHAPE 10
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass
  SELFWEIGHT X 1
  SELFWEIGHT Y 1
  SELFWEIGHT Z 1
JOINT LOAD
  17 TO 48 FX 7
  49 TO 64 FX 3.5
  17 TO 48 FY 7
  49 TO 64 FY 3.5
  17 TO 48 FZ 7
  49 TO 64 FZ 3.5
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3.5
DIA 2 TYPE RIG HEI 7
DIA 3 TYPE RIG HEI 10.5
*** Equivelant Lateral Force Definition ***
DEFINE NRC 2010 LOAD
  SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1.2 MVZ 1.2
  RDX 1.4 RDZ 3 ROX 1.5 ROZ 1.5 STX 3 STZ 4 MD 1
******************************************************************************
*** X-DIRECTION
LOAD 1 FX+TX
NRC LOAD X 1 DEC 1 ACC 0.1
Related Links

- [V. NRC 2010 Response Spectrum](on page 3563)

**TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994**

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 1994 edition of Eurocode 8 (EC8) for dynamic analysis.

**General Format**

```
SPECTRUM comb-method EURO {ELASTIC | DESIGN} {*} X f1 Y f2 Z f3} ACCELERATION
{DAMP f5} CDAMP | MDAMP } ( {LINEAR | LOGARITHMIC} ) (MISSING f6) (ZPA f7) ({ DOMINANT f10} SIGN }) (SAVE) (IMR f11) (STARTCASE f12)
```

**Note:** The data from SPECTRUM through ACC must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, the response spectra is input using following standard input parameters:

```
SOIL TYPE { A | B | C} ALPHA f8 Q f9
```

Unlike the custom defined response spectra, the EC 8 response spectra does not input use frequency-acceleration pairs. Based on the type of Response Spectra (Elastic/Design), Soil Type, Alpha, and Q, the program generates the applicable response spectra curve using the guidelines of section 4.2.2 or 4.2.4 of Eurocode 8 as applicable.
Where:

Table 261: Parameters used for Eurocode 8 1994 response spectrum

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X ( f_1 ), Y ( f_2 ), Z ( f_3 )</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>DAMP ( f_5 )</td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE Damping Information command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510)</td>
</tr>
<tr>
<td>MISSING ( f_6 )</td>
<td></td>
<td>Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the ( f_6 ) value entered in length/sec^2 (this value is not multiplied by SCALE). If ( f_6 ) is zero, then the spectral acceleration at the ZPA ( f_7 ) frequency is used. If ( f_7 ) is zero or not entered, the spectral acceleration at 33 Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. <strong>Note:</strong> If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.</td>
</tr>
<tr>
<td>ZPA ( f_7 )</td>
<td>33 [Hz]</td>
<td>The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING ( f_6 ) is entered.</td>
</tr>
<tr>
<td>DOMINANT ( f_{10} )</td>
<td>1 (1st Mode)</td>
<td>The dominant mode method. All results will have the same sign as mode number ( f_{10} ) alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). <strong>Note:</strong> Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</td>
</tr>
<tr>
<td>Parameter</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>---------------</td>
<td>------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>IMR f11</td>
<td>1</td>
<td>The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.</td>
</tr>
<tr>
<td>STARTCASE f12</td>
<td>Highest Load Case No. + 1</td>
<td>The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If f12 is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.</td>
</tr>
<tr>
<td>ALPHA f8</td>
<td></td>
<td>The design ground acceleration expressed in terms of acceleration due to gravity (g). For most of the application of Eurocode 8, the hazard is described in terms of a single parameter (i.e., the value of effective peak ground acceleration in rock or firm soil). This acceleration is termed as the design ground acceleration.</td>
</tr>
<tr>
<td>Q f9</td>
<td></td>
<td>The behavior factor used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.</td>
</tr>
</tbody>
</table>

**comb-method** = { SRSS | ABS | CQC | ASCE | TEN | CSM | GRP } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

**SRSS** Square Root of Summation of Squares method.

**ABS** Absolute sum. This method is very conservative and represents a worst case combination.

**CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \left( \sum_n \sum_m f_n \rho_{nm} f_m \right)^{1/2} \]

where

\[ \rho_{nm} = \frac{8 \xi^2 (1 + r) r^{2/3}}{(1 - r^2)^2 + 4 \xi^2 r(1 + r)^2} \]

\[ r = \frac{\omega_n}{\omega_m} \leq 1.0 \]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.
ASCE  NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


CSM  Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.


**Note:** If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

**ELASTIC or DESIGN**

The response spectrum loading can be based on either **Elastic** or **Design** response spectra. Refer to Eurocode 8.

The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor \( Q \) and the reduced response spectrum is termed as “Design Response Spectrum.” STAAD.Pro generates the Elastic Response Spectra using the guidelines of section 4.2.4 and Table 4.2 of Eurocode 8.

So, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra is to be used.

**ACCELERATION**

indicates that the Acceleration spectra will be entered.

**Note:** Eurocode 8 does not have provisions displacement response spectra.

**DAMP, MDAMP, and CDAMP**

select source of damping input:

- **DAMP** indicates to use the \( f_2 \) value for all modes
- **MDAMP** indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

**LINEAR or LOGARITHMIC**

Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When **FILE filename** is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.
SIGN
This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

Caution: Do not enter DOMINANT parameter with this option.

SAVE
This option results in the creation of an acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec². These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

SOIL TYPE
This parameter is used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of three kinds:

- Type A: for Rock or stiff deposits of sand
- Type B: for deep deposits of medium dense sand, gravel or medium stiff clays.
- Type C: Loose cohesionless soil deposits or deposits with soft to medium stiff cohesive soil.

Refer to section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

Individual Modal Response Case Generation

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688) for additional details on IMR load case generation.

Description
This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. Maximum response spectrum load cases allowed in one run is 4.
Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections TR.32 Loading Specifications (on page 2650) and G.17.3 Dynamic Analysis (on page 2362).

Multiple Response Spectra

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area are generated by sources varying widely in location and other parameters. In those cases different values of ALPHA as well as Q may be required to indicate the different shapes of response spectrum for each type of earthquake.

Example

```plaintext
LOAD 2 SPECTRUM X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM SRSS EURO ELASTIC X 1 ACC DAMP 0.05 –
LIN MIS 0 ZPA 40
SOIL TYPE A ALPHA 2 Q 1.5
```

Related Links

- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3 Damping Modeling (on page 2366)
- G.17.3 Dynamic Analysis (on page 2362)

TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2004 edition of Eurocode 8, “General Rules, seismic actions and rules for buildings”, BS EN 1998-1:2004. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method EURO 2004 {ELASTIC | DESIGN} {RS1 | RS2} *( X f1| Y f2| Z f3}
ACCELERATION
{DAMP f5| CDAMP | MDAMP } ( {LINEAR | LOGARITHMIC } ) (MISSING f6) (ZPA f7) ( { DOMINANT f10 | SIGN }) (SAVE) (IMR f11) (STARTCASE f12)
```

Note: The data SPECTRUM through ACC must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).
Starting on the next line, the response spectra is input using following standard input parameters:

\[
\text{SOIL TYPE \{ A | B | C | D | E \} ALPHA f8 Q f9}
\]

Unlike the custom defined response spectra the EC 8 response spectra is not input using frequency-acceleration pairs. Based on the type of Response Spectra (Elastic/Design), Soil Type, Alpha and Q the software generates the applicable response spectra curve using the guidelines of section 3.2.2.2 or 3.2.2.3 or 3.2.2.5 of Eurocode 8: 2004 as applicable.

Where:

Table 262: Parameters used for Eurocode 8 2004 response spectrum

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(X f1, Y f2, Z f3)</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>(DAMP f5)</td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If (CDAMP) is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If (MDAMP) is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510)</td>
</tr>
<tr>
<td>(MISSING f6)</td>
<td></td>
<td>Optional parameter to use &quot;Missing Mass&quot; method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the (f6) value entered in length/sec(^2) (this value is not multiplied by SCALE).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If (f6) is zero, then the spectral acceleration at the (ZPA f7) frequency is used. If (f7) is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> If the (MISSING) parameter is entered on any spectrum case it will be used for all spectrum cases.</td>
</tr>
<tr>
<td>(ZPA f7)</td>
<td>33 [Hz]</td>
<td>The zero period acceleration value for use with (MISSING) option only. Defaults to 33 Hz if not entered. The value is printed but not used if (MISSING \ f6) is entered.</td>
</tr>
</tbody>
</table>
## Technical Reference of STAAD Commands

**TR.32 Loading Specifications**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DOMINANT f1θ</strong></td>
<td>1 (1st Mode)</td>
<td>The dominant mode method. All results will have the same sign as mode number ( f_{1\theta} ) alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). <strong>Note:</strong> Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</td>
</tr>
<tr>
<td><strong>IMR f11</strong></td>
<td>1</td>
<td>The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted ( (NM) ), then it will be reset to NM. Modes one through ( f_{11} ) will be used. Missing Mass modes are not output.</td>
</tr>
<tr>
<td><strong>STARTCASE f12</strong></td>
<td>Highest Load Case No. + 1</td>
<td>The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If ( f_{12} ) is not higher than all prior load case numbers, then the default will be used. For modes 2 through ( NM ), the load case number is the prior case number plus one.</td>
</tr>
<tr>
<td><strong>ALPHA f1θ</strong></td>
<td></td>
<td>Alpha is the “Ground Acceleration On Type A Ground” and defined in Eurocode 8 as ( a_g ) as used in equations 3.2 – 3.5. Refer to the Eurocode for further information. <strong>Note:</strong> The specified alpha factor in STAAD.Pro is given as the ratio of the ground acceleration ( a_g ) to acceleration due to gravity. That is, the value for ( f_{1\theta} ) should be a factor of ( G ), not an actual acceleration ( e.g. ), if the value of ( a_g = 1.5 \text{ m/s}^2 ), then enter the ALPHA value of ( 1.5 \text{ m/s}^2 / 9.81 \text{ m/s}^2 = 0.15 ).</td>
</tr>
<tr>
<td><strong>Q f11</strong></td>
<td></td>
<td>Q is the ‘Behaviour Factor’ and is an approximation of the ratio of the seismic forces that the structure would experience if its response was completely elastic with 5% viscous damping, to the seismic forces that may be used in design, with a conventional elastic analysis model, still ensuring a satisfactory response of the structure.</td>
</tr>
</tbody>
</table>

\( \text{comb-method} = \{ \text{SRSS} \mid \text{ABS} \mid \text{CQC} \mid \text{ASCE} \mid \text{TEN} \mid \text{CSM} \mid \text{GRP} \} \) are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

**SRSS** Square Root of Summation of Squares method.

**ABS** Absolute sum. This method is very conservative and represents a worst case combination.
CQC  Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m} \]

where

\[ \rho_{nm} = \frac{8 \zeta^2 (1 + r) r^{2/3}}{(1 - r^2)^2 + 4 \zeta^2 r (1 + r)^2} \]

\[ r = \frac{\omega_n}{\omega_m} \leq 1.0 \]

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE  NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


CSM  Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.


Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

See G.17.3 Dynamic Analysis (on page 2362), TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2538), and TR.34 Frequency Calculation (on page 2789)

The keywords EURO 2004 is mandatory to denote that the applied loading is as per the guidelines of Eurocode 8.

ELASTIC or DESIGN  The response spectrum loading can be based on either Elastic or Design response spectra. Refer to Eurocode 8.

Design The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor $Q$ and the reduced response spectrum is termed as "Design Response Spectrum." STAAD.Pro generates the Elastic Response Spectra using the guidelines of section 4.2.4 and Table 4.2 of Eurocode 8.

So, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra is to be used.

RS1 or RS2  Two types of response spectra curve can be generated based on either response spectra type 1 curve (RS1) or response spectra type 2 curve (RS2).

ACCELERATION  indicates that the Acceleration spectra will be entered.
Note: Eurocode 8 does not have provisions displacement response spectra.

**DAMP, MDAMP, and CDAMP**

select source of damping input:

- **DAMP** indicates to use the $f_2$ value for all modes
- **MDAMP** indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

**LINEAR or LOGARITHMIC**

Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When **FILE filename** is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**SIGN**

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for **ABS** option.

**Caution:** Do not enter DOMINANT parameter with this option.

**SAVE**

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec$^2$. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

**SOIL TYPE**

This parameter is used to define the subsoil conditions based on which the response spectra will be generated as defined in Table 3.1 Ground Types. Based on the subsoil conditions the soil types may be of five kinds:

- **Type A**: rock or other rock-like geographical formation.
- **Type B**: very dense sand, gravel or very stiff clay.
- **Type C**: Deep deposits of dense or medium dense sand, gravel or stiff clay.
- **Type D**: Deposits of loose-to-medium cohesionless soil or of predominantly soft to firm cohesive soil.
- **Type E**: Surface alluvium layer

Please refer section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

**Individual Modal Response Case Generation**

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if
five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

**Note:** When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688) for additional details on IMR load case generation.

**Description**

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see TR.32 Loading Specifications (on page 2650) and G.17.3 Dynamic Analysis (on page 2362).

**Multiple Response Spectra**

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area is generated by sources varying widely in location and other parameters. In such cases different values of ALPHA as well as Q may be required to indicate the different shapes of response spectrum for each type of earthquake.

```
LOAD 2 SPECTRUM X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
  10 FX 17.5
  10 FY 17.5
  10 FZ 17.5
MEMBER LOADS
  5 CON GX 5.0 6.0
  5 CON GY 5.0 6.0
  5 CON GX 7.5 10.0
  5 CON GY 7.5 10.0
  5 CON GX 5.0 14.0
```
Related Links

- M. To add an EC8 response spectrum (on page 849)
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3 Dynamic Analysis (on page 2362)

**TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002**

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per IS: 1893 (Part 1)-2002 for dynamic analysis.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads.

**Note:** This facility has *not* been developed for cases where the Z axis is set to be the vertical direction using the SET Z UP command.

**General Format**

The data in the following format can be contained all on a single line or broken into two or three lines, so long as the second and third lines start with the ACC and DOMINANT or SIGN commands, respectively.

```
SPECTRUM comb-method 1893 (TORSION) (DECCENTRICITY f8) (ECCENTRICITY f9) *{ X f1 | Y f2 | Z f3 }
{ ACCELERATION | DISPLACEMENT } (SCALE f4) { DAMP f5 | CDAMP | MDAMP } (MISSING f6)
{ (DOMINANT f10 | SIGN }) (SAVE) (IMR f11) (STARTCASE f12)
```

The following command (SOIL TYPE parameter or response spectra data pairs) must be in a separate line.

```
{ SOIL TYPE f11 | *{ P1,V1; P2,V2; P3,V3;...PN,VN } }
```

The following command, if present, must be on a separate line. This performs the optional soft story check.

```
( CHECK SOFT STORY )
```

The following command, if present, must be on a separate line. This performs the story drift check.

```
( CHECK STORY DRIFT ) (RF f14)
```

Where:
Table 263: Parameters used for IS: 1893 (Part 1) 2002 response spectrum

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DECCENTRICITY $f_8$</td>
<td>1.0</td>
<td>(Optional input) It is a factor which when multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity) gives dynamic eccentricity. Since the applied load is acting at the center of mass, the effect of inherent torsion arising due to static eccentricity is included in the analysis. A factor greater than 1.0 is used to consider additional inherent torsion that will act at center of mass. Default value is 1.0.</td>
</tr>
<tr>
<td>Note:</td>
<td></td>
<td>The torsion arising due to dynamic eccentricity (i.e., static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.9.2 of the code. The dynamic eccentricity is automatically calculated by the program while you can specify the amount of accidental eccentricity (if not specified, the default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered). For details See Torsion Methodology (on page 2729).</td>
</tr>
<tr>
<td>ECCENTRICITY $f_9$</td>
<td>0.05</td>
<td>It is a factor which indicates the extent of accidental eccentricity. For all buildings this factor is to be provided as 0.05. However, for highly irregular buildings this factor may be increased to 0.10. This factor is to be externally provided to calculate design eccentricity. Since accidental eccentricity can be on either side, you must consider lateral force acting at a floor level to be accompanied by a clockwise or a counterclockwise accidental torsion moment. If $f_8$ value is positive, it indicates clockwise torsion whereas the negative value indicates counterclockwise torsion.</td>
</tr>
<tr>
<td>$X f_1, Y f_2, Z f_3$</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>SCALE $f_4$</td>
<td>1.0</td>
<td>Linear scale factor by which the spectra data will be multiplied. Usually used to factor g’s to length/sec$^2$ units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s$^2$ or 32.2 ft/s$^2$).</td>
</tr>
<tr>
<td>Parameter</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| DAMP $f_5$    | 0.05          | The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.  
If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)  
If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510) |
| MISSING $f_6$ |               | Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the $f_6$ value entered in length/sec$^2$ (this value is not multiplied by SCALE).  
If $f_6$ is zero, then the spectral acceleration at the ZPA $f_7$ frequency is used. If $f_7$ is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results.  
**Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases. |
| ZPA $f_7$     | 33 [Hz]       | The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING $f_6$ is entered. |
| IGNORE $f_{13}$ | 0.009        | (Optional input) It indicates the mass participation (in percent) of those modes to be excluded while considering torsion provision of IS-1893. Depending upon the model it may be found that there are many local modes and torsional modes whose mass participation is practically negligible. These modes can be excluded without much change in the final analysis result. If not provided all modes will be considered. If none provided the default value of 0.009% will be considered. If IGN is entered on any one spectrum case it will be used for all spectrum cases.  
**Note:** If the value of $f_{14}$ is considerable it may lead to considerable variation of analysis result from the actual one. Hence caution must be taken while using IGNORE command. |
<p>|               |               | If the MODE SELECT command is provided along with the IGNORE command, the number of modes excluded from the analysis will be those deselected by the MODE SELECT command and also those deselected by the IGNORE command. |</p>
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DOMINANT f10</td>
<td>1 (1st Mode)</td>
<td>The dominant mode method. All results will have the same sign as mode number f10 alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). <strong>Note:</strong> Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.</td>
</tr>
<tr>
<td>IMR f11</td>
<td>1</td>
<td>The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.</td>
</tr>
<tr>
<td>STARTCASE f12</td>
<td>Highest Load Case No. + 1</td>
<td>The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If f12 is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.</td>
</tr>
<tr>
<td>SOIL TYPE f11</td>
<td></td>
<td>The soil type present. Depending upon time period, types of soil and damping, average response acceleration coefficient, $S_a/g$ is calculated.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1 = for rocky or hard soil</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 = medium soil</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 = soft soil sites</td>
</tr>
<tr>
<td>custom P1,V1; P2,V2; P3,V3; ... Pn,Vn</td>
<td></td>
<td>Data is part of input immediately following spectrum command for a “custom” response spectrum. Period - Value pairs (pairs separated by semicolons) are entered to describe the spectrum curve. Period is in seconds and the corresponding Value is acceleration (current length unit/ sec^2). If data is in g acceleration units then the factor by which spectra data will be multiplied is g to the current length unit (9.81, 386.4, etc). <strong>Note:</strong> Do not enter if a SOIL TYPE f11 value is specified.</td>
</tr>
</tbody>
</table>
Parameter | Default Value | Description
--- | --- | ---
RF \( f14 \) | | The response reduction factor. If not specified, the program will look for the factor defined under DEFINE 1893 LOAD (refer to TR.31.2.9 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 257)). If none is provided there either, a factor of 1.0 is assumed. The response reduction factor represents ratio of maximum seismic force on a structure during specified ground motion if it were to remain elastic to the design seismic force. Actual seismic force is reduced by a factor RF to obtain design force.

1893 indicates the analysis as per IS:1893 (Part 1)-2002 procedures.

\( \text{comb-method} = \{ \text{SRSS} | \text{ABS} | \text{CQC} | \text{ASCE} | \text{TEN} | \text{CSM} | \text{GRP} \} \) are methods of combining the responses from each mode into a total response.

**Note:** CQC, SRSS, and CSM Grouping methods are recommended by IS:1893 (Part 1) –2002.

**SRSS** Square Root of Summation of Squares method.

**ABS** Absolute sum. This method is very conservative and represents a worst case combination.

**CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[
F = \sqrt{\sum_{n} \sum_{m} f_{n} \rho_{nm} f_{m}}
\]

where

\[
\rho_{nm} = \frac{8 \xi^{2}(1 + r) r^{2}/3}{(1 - r^{2})^{2} + 4 \xi^{2} r(1 + r) r^{2}}
\]

\[
r = \frac{\omega_{n}}{\omega_{m}} \leq 1.0
\]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.

**ASCE** NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


**CSM** Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

**GRP** Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

**TORSION** indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Torsion (on page 2729) for additional information.

**Note:** If TORSION is entered on any one spectrum case it will be used for all spectrum cases.
Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

**ACCELERATION or DISPLACEMENT**

Indicates whether **Acceleration** or **Displacement** spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

\[ \text{Displacement} = \text{Acceleration} \times (1/\omega)^2 \]

where

\[ \omega = \frac{2\pi}{\text{Period}} \text{ (period given in seconds; } \omega \text{ in cycles per second)} \]

**DAMP, MDAMP, and CDAMP**

Select source of damping input:
- **DAMP** indicates to use the \( f_2 \) value for all modes
- **MDAMP** indicates to use the damping entered or computed with the **DEFINE DAMP** command if entered, otherwise default value of 0.05 will be used
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode.

You must specify damping for different materials under the **CONSTANT** specification.

**SIGN**

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

**Caution:** Do not enter DOMINANT parameter with this option.

**SAVE**

This option results in the creation of an acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

**CHECK SOFT STORY**

Indicates that soft story checking will be performed. If omitted from input, there will be no soft story checking. Refer to **TR.28.2.1 Soft Story Checking** (on page 2529) for details.

**CHECK STORY DRIFT**

Indicates that a story drift check is to be performed.

**Tip:** This is done in place of post-analysis story drift checks for IS 1893-2002.

**Methodology**

The design lateral shear force at each floor in each mode is computed by STAAD.Pro in accordance with the Indian IS: 1893 (Part 1)-2002 equations 7.8.4.5c and 7.8.4.5d.

\[ Q_{ik} = A_k \cdot \phi_{ik} \cdot P_k \cdot W_k \]

and

\[ V_{ik} = \sum_{i=i+1}^{n} Q_{ik} \]

**Note:** All symbols and notations in the above equation are as per IS: 1893(Part 1)-2002.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads.
1. You provide the value for \( Z/2 \cdot I/R \) as factors for input spectrum. You calculate the expression \( Z/2 \cdot I/R \) and provide these values using the terms \( f_1, f_2, \) and \( f_3 \) and applicable, where these terms are explained in the table below.

2. The program calculates time periods for first six modes or as specified.

3. The program calculates \( Sa/g \) for each mode utilizing time period and damping for each mode.

4. The program calculates the design horizontal acceleration spectrum value \( A_k \) for each mode.

5. The program then calculates mode participation factor for each mode.

6. The peak lateral seismic force at each floor in each mode is calculated.

7. All response quantities for each mode are calculated.

8. The peak response quantities are then combined as per the specified method (SRSS, CQC, ABS, CSM or TEN) to get the final results.

*Individual Modal Response Case Generation*

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be \( \omega^2 \) times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

**Note:** When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.

Refer to TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688) for additional details on IMR load case generation.

**Notes**

a. The design base shear \( V_B \), calculated from the Response Spectrum method, is compared with the base shear \( V_b \), calculated by empirical formula for the fundamental time period. If \( V_B \) is less than \( V_b \), all of the response quantities are multiplied by \( V_b / V_B \) as per Clause 7.8.2.
For this, the following input is necessary before defining any primary load case.

```
DEFINE  1893 LOAD

ZONE f1 1893-spec
SELFWEIGHT

JOINT WEIGHT
joint-list WEIGHT w

MEMBER WEIGHT
...

UNI v1 v2 v3
mem-list

CON v4 v5
...

CHECK SOFT STORY
```

1893-Spec = \{RF f2, I f3, SS f4, (ST f5), DM f6, (PX f7), (PZ f8), (DT f9)\}

Refer to TR.31.2.9 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2576) for full details on this command structure.

**Note:** STAAD.Pro does not calculate the fundamental frequency of the structure needed for the empirical base shear \(V_b\) calculation; so you must enter either the ST parameter or the PX and PZ parameters in the DEFINE 1893 LOAD data.

**b.** The following interpolation formula is adopted for interpolation between damping values as given in Table 3.

Interpolation and/or extrapolation of ground response acceleration for a particular mode has been made for determining the spectrum ordinates corresponding to the modal damping value for use in Response Spectrum analysis. The relationship that shall be used for this purpose is defined by:

\[ S_a = A e^{-\xi} + B/\xi \]

Where:

- \( S_a \) = Spectrum ordinate
- \( \xi \) = damping ratio

Constants \( A \) and \( B \) are determined using two known spectrum ordinates \( S_{a1} \) and \( S_{a2} \) corresponding to damping ratios \( \xi_1 \) and \( \xi_2 \), respectively, for a particular time period and are as follows:

\[ A = \frac{S_{a1}\xi_1 - S_{a2}\xi_2}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}} \]

\[ B = \frac{\xi_1 \xi_2 (S_{a2} e^{-\xi_1} - S_{a1} e^{-\xi_2})}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}} \]

Where:
The story drift in any story shall not exceed 0.004 times the story height as per Clause 7.11.1. To check this, the following command should be given after the analysis command.

```
PRINT STOREY DRIFT
```

A warning message will be printed if story drift exceeds this limitation.

d. If any soft story (as per definition in Table 5 of IS:1893-2002) is detected, a warning message will be printed in the output.

**Torsion**

The torsion arising due to dynamic eccentricity (i.e., static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.9.2 of the IS 1893 code. The dynamic eccentricity is automatically calculated by the program (in both cases of TOR and TOR OPP options), while the amount of accidental eccentricity can be specified through the ECC option (if not specified, default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered).

Non-symmetric or torsionally unbalanced buildings are prone to earthquake damage due to coupled lateral and torsional movements (i.e., the translational vibration of the building couples with its torsional vibrations within elastic range). The level of coupling between lateral and torsional vibrations of the building can be larger, thus leading to significantly higher lateral-torsional coupling than that predicted by elastic analysis.

- Cl. 7.8.4.5 of IS 1893 (Part 1) : 2002 is valid for buildings with regular or nominally irregular plan configurations. For buildings which are irregular in plan, it is better to consider torsion from dynamic eccentricity into analysis; even if torsionally coupled vibration is considered during response spectrum analysis.
- Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 states that, in the case that a 3D dynamic analysis is carried out, the dynamic amplification factor 1.5—as given by Cl. 7.9.2—can be replaced by 1.0. This implies that the code also recommends to use Cl. 7.9.2 for all types of buildings by including torsion from both dynamic and accidental eccentricity in the response spectrum analysis.

**Torsion Methodology**

As per IS1893-2002 code, provision shall be made in all buildings for increase in shear forces on the lateral force resisting elements resulting from the horizontal torsional moment arising due to eccentricity between the center of mass and the center of rigidity.

In response spectrum analysis all the response quantities (i.e. joint displacements, member forces, support reactions, plate stresses, etc) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.
For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

**Steps**
For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated. Refer Cl. 7.8.4.5c of IS 1893 (Part 1) : 2002. \(Q_{ik}\) at floor \(i\) for mode \(k\)
2. At each floor design eccentricity is calculated. Refer Cl. 7.9.2 and Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 of IS 1893 (Part 1) : 2002.
   
   Thus, design eccentricity \(e_{di} = f_{15} \times e_{si} + f_{12} \times b_i\) where \(f_{15} = 1.0\) and \(f_{12} = (\pm) 0.05\)
   
   Where:
   
   \(e_{si}\) = dynamic eccentricity arising due to center of mass and center of rigidity at floor \(i\) (static eccentricity multiplied by dynamic amplification factor 1.0 for response spectrum analysis),
   
   \(b_i\) = floor plan dimension in the direction of earthquake loading.
3. Torsional moment is calculated at each floor. \(M_{ik} = Q_{ik} \times e_{di}\) at floor \(i\) for mode \(k\)
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

**Modal Combination**
Steps 1 to 6 are performed for all modes considered and missing mass correction (if any). Finally the peak response quantities from different modal response are combined as per CQC or SRSS or TEN PERCENT or CSM method.

**Notes**
After the analysis is complete following files are generated.

a. Story shear for each mode for each load case is given in the file \(<\text{filename}>\_RESP1893.txt\).

b. Rotational stiffness of each floor is given in the file \(<\text{filename}>\_ROT1893.txt\).

c. Center of mass, center of rigidity, design eccentricity at each floor level and additional shear due to torsion at each floor level for each mode for each load case is given in the file \(<\text{filename}>\_TOR1893.txt\).

**Dynamic Eccentricity**
The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

the rotational component of ground motion about the vertical axis,
the difference between computed and actual values of the mass, stiffness, or strength, and uneven live mass distribution.

In most country’s seismic codes, the provision for design eccentricity $e_{di}$ at $i^{th}$ floor of a building is given by the following equations:

$$e_{di} = \alpha e_{si} + \beta b_i$$
$$e_{di} = \delta e_{si} - \beta b_i$$

where

- $e_{si}$ = static eccentricity at $i^{th}$ floor
- $b_i$ = plan dimension of the $i^{th}$ floor normal to the direction of ground motion
- $\alpha$, $\beta$, and $\delta$ = specified constants

If $\alpha = 1.0$, $\delta = 1.0$, and $\beta = 0.05$ parameters to be defined are DEC = 1.0 and ACC = 0.05 in one load case and DEC = 1.0 and ACC = -0.05 in another load case to include both natural and accidental torsion.

Only TOR ECC 0.05 or TOR ECC -0.05 can also be defined without specifying DEC 1.0 since it is the default that is included in the analysis.

The cases (d) and (e) are followed i.e. seismic load is passed through CM instead of CR. Natural torsion is automatically included in analysis for DEC $\leq$ 1.0 i.e., no additional inherent torsion is applied. However, if DEC $>$ 1.0, a twisting moment with modified eccentricity of DEC - 1 will act at CM. In this case, a message occurs in the output.
Example
Refer to V. IS 1893 2002 Response Spectrum (on page 3478) for a detailed explanation of this example.

LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS 1893 X 0.036 ACC DAMP 0.05
SOIL TYPE 1

Related Links
• M. To add an IS 1893 response spectrum (on page 846)
• G.17.3.3 Damping Modeling (on page 2366)
• G.17.3.3 Damping Modeling (on page 2366)
• G.17.3 Dynamic Analysis (on page 2362)
• V. IS 1893 2002 Response Spectrum (on page 3478)

TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2016
This command may be used to specify and apply the RESPONSE SPECTRUM loading as per IS: 1893 (Part 1)-2016 for dynamic analysis.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads.

Note: This facility has not been developed for cases where the Z axis is set to be the vertical direction using the SET Z UP command.

General Format
The data in the following format can be contained all on a single line or broken into two or three lines, so long as the second and third lines start with the ACC and DOMINANT or SIGN commands.

SPECTRUM comp-method IS1893 2016 (TORSION) (DECCENTRICITY f8) (ECCENTRICITY f9) *{ X
f1 | Y f2 | Z f3 }

{ ACCELERATION | DISPLACEMENT } (SCALE f4) {DAMP f5 | CDAMP | MDAMP } ( { LINEAR |
LOGARITHMIC } ) (MISSING f6) (ZPA f7) (IGNORE f13)
{ { DOMINANT f10 | SIGN }} (IMR f11) (STARTCASE f12)

The following command (SOIL TYPE parameter or response spectra data pairs) must be in a separate line.

{ SOIL TYPE f11 | *{ P1,V1; P2,V2; P3,V3;...PN,VN } }

Note: The spectrum type options ACCELERATION or DISPLACEMENT should only be used with custom soil types for IS 1893 2016.

The following command, if present, must be on a separate line. This performs the story drift check.

( CHECK STORY DRIFT ) (RF f14)

Where:
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DECCENTRICITY ( f_8 )</td>
<td>1.0</td>
<td>(Optional input) It is a factor which when multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity) gives dynamic eccentricity. Since the applied load is acting at the center of mass, the effect of inherent torsion arising due to static eccentricity is included in the analysis. A factor greater than 1.0 is used to consider additional inherent torsion that will act at center of mass. Default value is 1.0.</td>
</tr>
<tr>
<td>ECCENTRICITY ( f_9 )</td>
<td>0.05</td>
<td>It is a factor which indicates the extent of accidental eccentricity. For all buildings this factor is to be provided as 0.05. However, for highly irregular buildings this factor may be increased to 0.10. This factor is to be externally provided to calculate design eccentricity. Since accidental eccentricity can be on either side, you must consider lateral force acting at a floor level to be accompanied by a clockwise or a counterclockwise accidental torsion moment. If ( f_8 ) value is positive, it indicates clockwise torsion whereas the negative value indicates counterclockwise torsion.</td>
</tr>
<tr>
<td>X ( f_1 ), Y ( f_2 ), Z ( f_3 )</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>SCALE ( f_4 )</td>
<td>1.0</td>
<td>Linear scale factor by which the spectra data will be multiplied. Usually used to factor g’s to length/sec(^2) units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s(^2) or 32.2 ft/s(^2)).</td>
</tr>
<tr>
<td>Parameter</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>DAMP $f_5$</td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to <a href="#">TR.26.2 Specifying Constants for Members and Elements</a> (on page 2503). If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to <a href="#">TR.26.4 Modal Damping Information</a> (on page 2510).</td>
</tr>
<tr>
<td>MISSING $f_6$</td>
<td></td>
<td>Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the $f_6$ value entered in length/sec$^2$ (this value is not multiplied by SCALE). If $f_6$ is zero, then the spectral acceleration at the ZPA $f_7$ frequency is used. If $f_7$ is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. <strong>Note:</strong> If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.</td>
</tr>
<tr>
<td>ZPA $f_7$</td>
<td>33 [Hz]</td>
<td>The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING $f_6$ is entered.</td>
</tr>
<tr>
<td>IGNORE $f_{13}$</td>
<td>0.009</td>
<td>(Optional input) It indicates the mass participation (in percent) of those modes to be excluded while considering torsion provision of IS-1893. Depending upon the model it may be found that there are many local modes and torsional modes whose mass participation is practically negligible. These modes can be excluded without much change in the final analysis result. If not provided all modes will be considered. If none provided the default value of 0.009% will be considered. If IGN is entered on any one spectrum case it will be used for all spectrum cases. <strong>Note:</strong> If the value of $f_{14}$ is considerable it may lead to considerable variation of analysis result from the actual one. Hence caution must be taken while using IGNORE command. If the MODE SELECT command is provided along with the IGNORE command, the number of modes excluded from the analysis will be those deselected by the MODE SELECT command and also those deselected by the IGNORE command.</td>
</tr>
<tr>
<td>Parameter</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>---------------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| DOMINANT $f10$    | 1 (1st Mode)  | The dominant mode method. All results will have the same sign as mode number $f10$ alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).  
**Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode. |
| IMR $f11$         | 1             | The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through $f11$ will be used. Missing Mass modes are not output. |
| STARTCASE $f12$   | Highest Load Case No. + 1 | The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If $f12$ is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one. |
| SOIL TYPE $f11$   |                | The soil type present. Depending upon time period, types of soil and damping, average response acceleration coefficient, $S_a/g$ is calculated.  
1 = for rocky or hard soil  
2 = medium soil  
3 = soft soil sites |
| $custom$ $P1,V1;$ | $P2,V2;$ $P3,V3;$ ...  | Data is part of input immediately following spectrum command for a “custom” response spectrum. Period - Value pairs (pairs separated by semicolons) are entered to describe the spectrum curve. Period is in seconds and the corresponding Value is acceleration (current length unit/ sec$^2$). If data is in g acceleration units then the factor by which spectra data will be multiplied is g to the current length unit (9.81, 386.4, etc).  
**Note:** Do not enter if a SOIL TYPE $f11$ value is specified.  
$Pn,Vn$ |
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RF f14</td>
<td></td>
<td>The response reduction factor. If not specified, the program will look for the factor defined under DEFINE 1893 2016 LOAD (refer to TR.31.2.10 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2582)). If none is provided there either, a factor of 1.0 is assumed. The response reduction factor represents ratio of maximum seismic force on a structure during specified ground motion if it were to remain elastic to the design seismic force. Actual seismic force is reduced by a factor RF to obtain design force.)</td>
</tr>
</tbody>
</table>

1893 indicates the analysis as per IS:1893(Part 1)-2016 procedures.

*comb-method* = { SRSS | ABS | CQC | ASCE | TEN | CSM | GRP } are methods of combining the responses from each mode into a total response.

**Note:** CQC, SRSS, and CSM Grouping methods are recommended by IS:1893 (Part 1) –2016.

**SRSS** Square Root of Summation of Squares method.

**ABS** Absolute sum. This method is very conservative and represents a worst case combination.

**CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \left( \sum_n \sum_m f_n \rho_{nm} f_m \right)^{\frac{1}{2}} \]

where

\[ \rho_{nm} = \frac{8\xi^2(1 + r) r^{2/3}}{(1 - r^2)^2 + 4\xi^2 r(1 + r)^2} \quad \omega_n/\omega_m \leq 1.0 \]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.

**ASCE** NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


**CSM** Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

**GRP** Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

**TORSION** indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Torsion (on page 2740) for additional information.

**Note:** If TORSION is entered on any one spectrum case it will be used for all spectrum cases.
Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

**ACCELERATION or DISPLACEMENT**

indicates whether Acceleration or Displacement spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

\[
\text{Displacement} = \text{Acceleration} \times (1/\omega)^2
\]

where

\[
\omega = \frac{2\pi}{\text{Period}} \text{ (period given in seconds; } \omega \text{ in cycles per second)}
\]

**DAMP, MDAMP, and CDAMP**

select source of damping input:

- DAMP indicates to use the \(f_2\) value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode.

You must specify damping for different materials under the CONSTANT specification

**LINEAR or LOGARITHMIC**

Select Linear or Logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**Note:** The LINEAR or LOGARITHMIC option can only be used for custom soil types (e.g., when response spectra data pairs are specified). Do not use these commands when the SOIL TYPE command is used.

**SIGN**

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

**Caution:** Do not enter DOMINANT parameter with this option.

**SAVE**

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec\(^2\). These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

**Note:** In order to perform a soft story check per IS 1893 2016, use the TR.28.2.1 Soft Story Checking (on page 2529) command.

**Methodology**

The design lateral shear force at each floor in each mode is computed by STAAD.Pro in accordance with the Indian IS: 1893 (Part 1)-2016 equations 7.8.4.5c and 7.8.4.5d.

\[
Q_{ik} = A_k \cdot \phi_{ik} \cdot P_k \cdot W_k
\]
and

\[ V_{ik} = \sum_{i=i+1}^{n} Q_{ik} \]

**Note:** All symbols and notations in the above equation are as per IS: 1893(Part 1)-2016.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads.

1. You provide the value for \( Z/2 \cdot I/R \) as factors for input spectrum. You calculate the expression \( Z/2 \cdot I/R \) and provide these values using the terms \( f_1, f_2, \) and \( f_3 \) and applicable, where these terms are explained in the table below.
2. The program calculates time periods for first six modes or as specified.
3. The program calculates \( Sa/g \) for each mode utilizing time period and damping for each mode, based on IS 1893 2016 Clause 6.4.2 and Fig. 2(b).
   For the Y direction, \( Sa/g \) is always equal to 2.5 per Clause 6.4.6.
4. The program calculates the design horizontal acceleration spectrum value \( A_k \) for each mode.
5. The program then calculates mode participation factor for each mode.
6. The peak lateral seismic force at each floor in each mode is calculated.
7. All response quantities for each mode are calculated.
8. The peak response quantities are then combined as per the specified method (SRSS, CQC, ABS, CSM or TEN) to get the final results.

According to clause 7.7.3b, the scale factor for the earthquake in the vertical direction shall be taken as the maximum of the scale factors in both the X and Z direction (see also Notes below). There are broadly three cases which the program considers to evaluate this maximum. These are load lists defined containing one of the following cases

i. response spectra in X, response spectra in Y, and response spectra in Z separately in load cases in any order
ii. response spectra in XY, response spectra in YZ, and response spectra in XYZ in individual load cases in any order
iii. a mix of the first two cases in any order

For a scale factor for any load case with a vertical component (that is, response spectra of Y, XY, YZ, or XYZ types), the program will use the maximum of the scale factor for all preceding load cases that are listed after a previous load case with a Y component. For example, for model with the following 11 load cases, the following scale factors will be used:

<table>
<thead>
<tr>
<th>Load Case No.</th>
<th>Type</th>
<th>Scale Factor</th>
<th>Load Case Used for Scale Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X</td>
<td>2.18</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>XY</td>
<td>2.18</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>YZ</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>4</td>
<td>XZ</td>
<td>2.18</td>
<td>4</td>
</tr>
<tr>
<td>5</td>
<td>Y</td>
<td>2.18</td>
<td>max(3,4) = 4</td>
</tr>
<tr>
<td>6</td>
<td>X</td>
<td>2.18</td>
<td>6</td>
</tr>
<tr>
<td>7</td>
<td>Z</td>
<td>1</td>
<td>7</td>
</tr>
<tr>
<td>8</td>
<td>Y</td>
<td>2.18</td>
<td>max(6,7) = 6</td>
</tr>
<tr>
<td>Load Case No.</td>
<td>Type</td>
<td>Scale Factor</td>
<td>Load Case Used for Scale Factor</td>
</tr>
<tr>
<td>--------------</td>
<td>------</td>
<td>--------------</td>
<td>-------------------------------</td>
</tr>
<tr>
<td>9</td>
<td>XY</td>
<td>2.18</td>
<td>9</td>
</tr>
<tr>
<td>10</td>
<td>XZ</td>
<td>2.18</td>
<td>10</td>
</tr>
<tr>
<td>11</td>
<td>Y</td>
<td>2.18</td>
<td>max(9,10) = 9 = 10</td>
</tr>
</tbody>
</table>

**Individual Modal Response Case Generation**

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

**Note:** When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.

Refer to TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688) for additional details on IMR load case generation.

**Notes**

**a.** The design base shear, $VB_{RS}$, calculated from the response spectrum method, is compared with the base shear, $VB_{SS}$, calculated by the empirical formula for the fundamental time period based on Clause 7.2.1. If $VB_{RS}$ is less than $VB_{SS}$, all of the response quantities are multiplied by $VB_{SS}/VB_{RS}$ as per Clause 7.7.3(a) for each of the orthogonal directions and by $\max\left[\left(\frac{VB_{SS}}{VB_{RS}}\right)x, \left(\frac{VB_{SS}}{VB_{RS}}\right)z\right]$ for response amplification when considering response spectrum load in the vertical direction based on Clause 7.7.3(b).

For this, the following input is necessary before defining any primary load case.

```plaintext
DEFINE IS1893 2016 LOAD
...
...
```

Refer to TR.31.2.10 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2582) for full details on this command structure.
Note: STAAD.Pro does not calculate the fundamental frequency of the structure needed for the empirical base shear \( VB_{SS} \) calculation; so you must enter either the \( ST \) parameter or the \( PX \) and \( PZ \) parameters in the \texttt{DEFINE IS1893 2016 LOAD} data.

b. Although IS 1893 2016 recommends a damping ratio of 0.05, you may use any other damping and the values will be calculated based on Table 3. A warning will be printed in the output.

c. The story drift in any story shall not exceed 0.004 times the story height as per Clause 7.11.1. To check this, the following command should be given after the analysis command:

\texttt{PRINT STORY DRIFT}

A warning message will be printed if the story drift exceeds this limitation.

d. If any soft story (as per definition in Table 5 of IS:1893-2016) is detected, a warning message will be printed in the output.

\textbf{Torsion}

The torsion arising due to dynamic eccentricity (i.e., static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.8 of the IS 1893 2016 code. The dynamic eccentricity is automatically calculated by the program (in both cases of \texttt{TOR} and \texttt{TOR OPP} options), while the amount of accidental eccentricity can be specified through the \texttt{ECC} option (if not specified, default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered).

Non-symmetric or torsionally unbalanced buildings are prone to earthquake damage due to coupled lateral and torsional movements (i.e., the translational vibration of the building couples with its torsional vibrations within elastic range). The level of coupling between lateral and torsional vibrations of the building can be larger, thus leading to significantly higher lateral-torsional coupling than that predicted by elastic analysis.

- Cl. 7.7.5.4 of IS 1893 (Part 1) : 2016 is valid for buildings with regular or nominally irregular plan configurations. For buildings which are irregular in plan, it is better to consider torsion from dynamic eccentricity into analysis; even if torsionally coupled vibration is considered during response spectrum analysis.

- Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 states that, in the case that a 3D dynamic analysis is carried out, the dynamic amplification factor 1.5—as given by Cl. 7.9.2—can be replaced by 1.0. This implies that the code also recommends to use Cl. 7.9.2 for all types of buildings by including torsion from both dynamic and accidental eccentricity in the response spectrum analysis.

\textbf{Torsion Methodology}

As per IS1893-2016 code, provision shall be made in all buildings for increase in shear forces on the lateral force resisting elements resulting from the horizontal torsional moment arising due to eccentricity between the center of mass and the center of rigidity.

In response spectrum analysis all the response quantities (i.e. joint displacements, member forces, support reactions, plate stresses, etc) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider
two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

**Steps**

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated. Refer Cl. 7.7.5.4(c) of IS 1893 (Part 1) : 2016. \( Q_{ik} \) at floor i for mode k

2. At each floor design eccentricity is calculated. Refer Cl. 7.9.2 and Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 of IS 1893 (Part 1) : 2016.

   Thus, design eccentricity \( e_{di} = f_{15} \times e_{si} + f_{12} \times b_i \) where \( f_{15} = 1.0 \) and \( f_{12} = (\pm) 0.05 \)

   Where:

   \( e_{di} \) = dynamic eccentricity arising due to center of mass and center of rigidity at floor i (static eccentricity multiplied by dynamic amplification factor 1.0 for response spectrum analysis),

   \( b_i \) = floor plan dimension in the direction of earthquake loading.

3. Torsional moment is calculated at each floor. \( M_{ik} = Q_{ik} \times e_{di} \) at floor i for mode k

4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.

5. Static analysis of the structure is performed with these nodal forces.

6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

**Modal Combination**

Steps 1 to 6 are performed for all modes considered and missing mass correction (if any). Finally the peak response quantities from different modal response are combined as per CQC or SRSS or TEN_PERCENT or CSM method.

**Notes**

After the analysis is complete following files are generated.

a. Story shear for each mode for each load case is given in the file <filename>_RESP1893.txt.

b. Rotational stiffness of each floor is given in the file <filename>_ROT1893.txt.

c. Center of mass, center of rigidity, design eccentricity at each floor level and additional shear due to torsion at each floor level for each load case is given in the file <filename>_TOR1893.txt.
Dynamic Eccentricity

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

In most country’s seismic codes, the provision for design eccentricity \( e_{di} \) at \( i^{th} \) floor of a building is given by the following equations:

\[
e_{di} = \alpha e_{si} + \beta b_i
\]

\[
e_{di} = \delta e_{si} - \beta b_i
\]

where

- \( e_{si} \) = static eccentricity at \( i^{th} \) floor
- \( b_i \) = plan dimension of the \( i^{th} \) floor normal to the direction of ground motion
- \( \alpha, \beta, \) and \( \delta \) = specified constants

If \( \alpha = 1.0, \delta = 1.0, \) and \( \beta = 0.05 \) parameters to be defined are \( DEC = 1.0 \) and \( ACC = 0.05 \) in one load case and \( DEC = 1.0 \) and \( ACC = -0.05 \) in another load case to include both natural and accidental torsion.

Only TOR ECC \( 0.05 \) or TOR ECC \( -0.05 \) can also be defined without specifying \( DEC = 1.0 \) since it is the default that is included in the analysis.

The cases (d) and (e) are followed i.e. seismic load is passed through CM instead of CR. Natural torsion is automatically included in analysis for \( DEC \leq 1.0 \) i.e., no additional inherent torsion is applied. However, if \( DEC >

### Diagrams

- **(a)** Design case,
- **(b)** Pure translation case,
- **(c)** Twisting moment including additional torsional moment plus accidental torsion,
- **(d)** Load applied through CM,
- **(e)** Twisting moment including additional torsional moment (if any) plus accidental torsion
1.0, a twisting moment with modified eccentricity of DEC-1 will act at CM. In this case, a message occurs in the output.

**Example**

Refer to *V. IS 1893 2016 Response Spectrum* (on page 3487) for a detailed explanation of this example.

```
LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS IS1893 2016 X 0.0432 DAMP 0.05
SOIL TYPE 1
```

**Related Links**

- **TR.31.2.14 IBC 2015 Seismic Load Definition** (on page 2600)
- **V. IS 1893 2016 Response Spectrum** (on page 3487)

**TR.32.10.1.8 Response Spectrum Specification per IBC 2006**

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2006 edition of the ICC specification *International Building Code* (IBC), for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

**General Format**

```
SPECTRUM comb-method IBC 2006 *{ X f1 | Y f2 | Z f3 } ACCELERATION
{DAMP f5 | CDAMP | MDAMP } ( { LINEAR | LOGARITHMIC } ) (MISSING f6) (ZPA f7)
{ { DOMINANT f10 | SIGN } } (SAVE) (IMR f11) (STARTCASE f12)
```

**Note:** The data from SPECTRUM through ACC must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

```
{ZIP f8 | LAT f9 LONG f13 | SS f14 S1 f15} SITE CLASS (f16) (FA f17 FV f18) TL f19
```

Where:

**Table 265: Parameters used for IBC 2006 response spectrum**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X f1, Y f2, Z f3</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>Parameter</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>---------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| DAMP \(f_5\) | 0.05          | The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.                                                                                                      
|             |               | If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)         
|             |               | If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510) |
| MISS\ing \(f_6\) |               | Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the \(f_6\) value entered in length/sec\(^2\) (this value is not multiplied by SCALE). \n|             |               | If \(f_6\) is zero, then the spectral acceleration at the ZPA \(f_7\) frequency is used. If \(f_7\) is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. |
|             |               | **Note:** If the MISS\ing parameter is entered on any spectrum case it will be used for all spectrum cases.                                                                                                                                           |
| ZPA \(f_7\) | 33 [Hz]       | The zero period acceleration value for use with MISS\ing option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISS\ing \(f_6\) is entered.                                                                 |
| DOM\inant \(f_{10}\) | 1 (1st Mode) | The dominant mode method. All results will have the same sign as mode number \(f_{10}\) alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). |
|             |               | **Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.                                                                                                                                       |
| IMR \(f_{11}\) | 1             | The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through \(f_{11}\) will be used. Missing Mass modes are not output. |
| STARTCASE \(f_{12}\) | Highest Load Case No. + 1 | The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If \(f_{12}\) is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one. |
## Technical Reference of STAAD Commands

### TR.32 Loading Specifications

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ZIP $f_8$</td>
<td>The zip code of the site location to determine the latitude and longitude and consequently the $S_s$ and $S_1$ factors. (IBC 2006, ASCE 7-02 Chapter 22)</td>
<td></td>
</tr>
<tr>
<td>LAT $f_9$</td>
<td>The latitude of the site used with the longitude to determine the $S_s$ and $S_1$ factors. (IBC 2006, ASCE 7-02 Chapter 22)</td>
<td></td>
</tr>
<tr>
<td>LONG $f_{13}$</td>
<td>The longitude of the site used with the latitude to determine the $S_s$ and $S_1$ factors. (IBC 2006, ASCE 7-02 Chapter 22)</td>
<td></td>
</tr>
<tr>
<td>SS $f_{14}$</td>
<td>Mapped MCE for 0.2s spectral response acceleration. (IBC 2006, ASCE 7-02 Chapter 22)</td>
<td></td>
</tr>
<tr>
<td>S1 $f_{15}$</td>
<td>Mapped spectral acceleration for a 1-second period. (IBC 2000, Section 16-17. IBC 2003, ASCE 7-02 section 9.4.1.2.4-2. IBC 2006, ASCE 7-05 Section 11.4.1)</td>
<td></td>
</tr>
<tr>
<td>SITE CLASS $f_{16}$</td>
<td>Enter A through F for the Site Class as defined in the IBC code. (IBC 2000, Section 1615.1.1 page 350. IBC 2003, Section 1615.1.1 page 322. IBC 2006 ASCE 7-05 Section 20.3)</td>
<td></td>
</tr>
<tr>
<td>FA $f_{17}$</td>
<td>Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)</td>
<td></td>
</tr>
<tr>
<td>FV $f_{18}$</td>
<td>Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)</td>
<td></td>
</tr>
<tr>
<td>TL $f_{19}$</td>
<td>Long-Period transition period in seconds. (IBC 2006, ASCE 7-02 Chapter 22)</td>
<td></td>
</tr>
</tbody>
</table>

**IBC 2006** indicates that the spectrum should be calculated as defined in the IBC 2006 specification.

**comb-method** = \{ SRSS | ABS | CQC | ASCE | TEN | CSM | GRP \} are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

**SRSS** Square Root of Summation of Squares method.

**ABS** Absolute sum. This method is very conservative and represents a worst case combination.

**CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$ F = \sqrt{\sum_n \sum_m F_n^2 F_m^2} $$

**STAAD.Pro** 2745  
User Manual
where
\[ \rho_{nm} = \frac{8\xi^2(1 + r)_{r}^{2/3}}{(1 - r^2)^2 + 4\xi^2r(1 + r)^2} \]
\[ r = \frac{\pi n}{\omega_n / \omega_m \leq 1.0} \]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.

**ASCE**  NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


**CSM**  Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

**GRP**  Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

**Note:** If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

**ACCELERATOIN**  indicates that the Acceleration spectra will be entered.

**Note:** IBC / ASCE 7 does not have provisions for displacement response spectra.

**DAMP, MDAMP, and CDAMP**
select source of damping input:
- **DAMP** indicates to use the \( f_2 \) value for all modes
- **MDAMP** indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

**LINEAR or LOGARITHMIC**
Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When **FILE filename** is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**SIGN**
This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for **ABS** option.

**Caution:** Do not enter **DOMINANT** parameter with this option.
SAVE

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Methodology
The methodology for calculating the response spectra is defined in ASCE7-05, section 11.4. The following is a quick summary:

a. Input $S_a$ and $S_1$ (this could have been searched from database or entered explicitly)

b. Calculate

$$S_{ms} = F_a \times S_a$$

and

$$S_{m1} = F_v \times S_1$$

Where:

$F_a$ and $F_v$ are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for $F_a$ and $F_v$ in lieu of table values.

c. Calculate

$$S_{ds} = \left(\frac{2}{3}\right) S_{ms}$$

and

$$S_{d1} = \left(\frac{2}{3}\right) S_{m1}$$

The spectrum is generated as per section 11.4.5.

Individual Modal Response Case Generation

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2770) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2795) & TR.38 Change Specification (on page 2835) must be entered after each such Spectrum case.
See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688) for additional details on IMR load case generation.

```
Example
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 6 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
...
LOAD 1 LOADTYPE Seismic TITLE RS_X
SPECTRUM SRSS IBC 2006 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
```

Related Links
- M. To add an IBC 2006 response spectrum (on page 848)
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3.3 Damping Modeling (on page 2366)

TR.32.10.1.9 Response Spectrum Specification per IBC 2012

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2012 edition of the ICC specification International Building Code (IBC) and ASCE 7-10, for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

**General Format**

```
SPECTRUM comb-method IBC 2012 (TORSION) (DECCENTRICITY f20) (ECCENTRICITY f21) {*{ X f1
| Y f2 | Z f3 } ACCELERATION

{DAMP f5 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f6) (ZPA f7) ({ DOMINANT f10 | SIGN })
(SAVE) (IMR f11) (STARTCASE f12)
```

**Note:** The data from SPECTRUM through ACC must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

```
{ZIP f8 | LAT f9 LONG f13 | SS f14 S1 f15 } SITE CLASS (f16) (FA f17 FV f18) TL f19
```

Where:

**Table 266: Parameters used for IBC 2012 response spectrum**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DECCENTRICITY f20</td>
<td>1.0</td>
<td>Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).</td>
</tr>
<tr>
<td>ECCENTRICITY f21</td>
<td>0.05</td>
<td>Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.</td>
</tr>
<tr>
<td>Parameter</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>X $f_1$, Y $f_2$, Z $f_3$</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>DAMP $f_5$</td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.</td>
</tr>
</tbody>
</table>

- If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)
- If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510)

| MISSING $f_6$ | Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the $f_6$ value entered in length/sec$^2$ (this value is not multiplied by SCALE). |

- If $f_6$ is zero, then the spectral acceleration at the ZPA $f_7$ frequency is used. If $f_7$ is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results.

**Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.

| ZPA $f_7$ | 33 [Hz] | The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING $f_6$ is entered. |

| DOMINANT $f_{10}$ | 1 (1st Mode) | The dominant mode method. All results will have the same sign as mode number $f_{10}$ alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). |

**Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.

| IMR $f_{11}$ | 1 | The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through $f_{11}$ will be used. Missing Mass modes are not output. |
### Technical Reference of STAAD Commands

#### TR.32 Loading Specifications

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>STARTCASE</strong> $f_{12}$</td>
<td>Highest Load Case No. + 1</td>
<td>The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If $f_{12}$ is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.</td>
</tr>
<tr>
<td><strong>ZIP</strong> $f_{11}$</td>
<td></td>
<td>The zip code of the site location to determine the latitude and longitude and consequently the $S_s$ and $S_1$ factors. (IBC 2012, ASCE 7-10 Chapter 22)</td>
</tr>
<tr>
<td><strong>LAT</strong> $f_{12}$</td>
<td></td>
<td>The latitude of the site used with the longitude to determine the $S_s$ and $S_1$ factors. (IBC 2012, ASCE 7-10 Chapter 22)</td>
</tr>
<tr>
<td><strong>LONG</strong> $f_{13}$</td>
<td></td>
<td>The longitude of the site used with the latitude to determine the $S_s$ and $S_1$ factors. (IBC 2012, ASCE 7-10 Chapter 22)</td>
</tr>
<tr>
<td><strong>SS</strong> $f_{14}$</td>
<td></td>
<td>Mapped MCE for 0.2s spectral response acceleration. (IBC 2012, ASCE 7-10 Chapter 22)</td>
</tr>
<tr>
<td><strong>S1</strong> $f_{15}$</td>
<td></td>
<td>Mapped spectral acceleration for a 1-second period. (IBC 2000, equation 16-17. IBC 2003, ASCE 7-02 section 9.4.1.2.4-2. IBC 2006, ASCE 7-05 Section 11.4.1)</td>
</tr>
<tr>
<td><strong>SITE CLASS</strong> $f_{16}$</td>
<td></td>
<td>Enter A through F for the Site Class as defined in the IBC code. (IBC 2000, Section 1615.1.1 page 350. IBC 2003, Section 1615.1.1 page 322. IBC 2006 ASCE 7-05 Section 20.3)</td>
</tr>
<tr>
<td><strong>FA</strong> $f_{17}$</td>
<td></td>
<td>Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)</td>
</tr>
<tr>
<td><strong>FV</strong> $f_{18}$</td>
<td></td>
<td>Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)</td>
</tr>
<tr>
<td><strong>TL</strong> $f_{19}$</td>
<td></td>
<td>Long-Period transition period in seconds. (IBC 2012, ASCE 7-10 Chapter 22)</td>
</tr>
</tbody>
</table>

**comb-method** = { SRSS | ABS | CQC | ASCE | TEN | CSM | GRP } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

- **SRSS** Square Root of Summation of Squares method.
- **ABS** Absolute sum. This method is very conservative and represents a worst case combination.
CQC  Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m} \]

where

\[ \rho_{nm} = \frac{\delta \zeta^2 (1 + r)^2 / 3}{(1 - r^2)^2 + 4 \zeta^2 r (1 + r)^2} \]

\[ r = \frac{\omega_n}{\omega_m} \leq 1.0 \]

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE  NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


CSM  Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.


Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

IBC 2012 indicates that the spectrum should be calculated as defined in the IBC 2012 specification.

TORSION  indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See “Torsion” for additional information.

Note: If TORSION is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

ACCELERATION  indicates that the Acceleration spectra will be entered.

Note: IBC / ASCE 7 does not have provisions for displacement response spectra.

DAMP, MDAMP, and CDAMP  select source of damping input:

- DAMP indicates to use the \( f_2 \) value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification
**Technical Reference of STAAD Commands**

**TR.32 Loading Specifications**

**LINEAR or LOGARITHMIC**

Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When **FILE filename** is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**SIGN**

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for **ABS** option.

**Caution:** Do not enter **DOMINANT** parameter with this option.

**SAVE**

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

**Methodology**

The methodology for calculating the response spectra is defined in ASCE 7-2010, section 11.4. The following is a quick summary:

a. Input $S_s$ and $S_1$ (this could have been searched from database or entered explicitly)

b. Calculate

$$S_{ms} = F_a \times S_s$$

and

$$S_{m1} = F_v \times S_1$$

Where:

$F_a$ and $F_v$ are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for $F_a$ and $F_v$ in lieu of table values.

c. Calculate

$$S_{ds} = \frac{2}{3} S_{ms}$$

and

$$S_{d1} = \frac{2}{3} S_{m1}$$

The spectrum is generated as per section 11.4.5.

See **TR.32.10.1.1 Response Spectrum Specification - Custom** (on page 2688) for additional details on IMR load case generation.
Torsion

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc.) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.
   
   Thus, design eccentricity \( e_{di} = f20 \times e_{si} + f12 \times b_i \) where \( f20 = 1.0 \) and \( f21 = (\pm) 0.05 \)

   Where:
   
   \( e_{si} \) = static eccentricity between center of mass and center of rigidity at floor \( i \).
   
   \( b_i \) = floor plan dimension orthogonal to the direction of earthquake loading.

3. Torsional moment is calculated at each floor.
   
   \( M_{ik} = Q_{ik} \times e_{di} \) at floor \( i \) for mode \( k \)

4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.

5. Static analysis of the structure is performed with these nodal forces.

6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)
Example

Refer to V. IBC 2012 Response Spectrum (on page 3422) for a detailed explanation of this example.
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS

LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS IBC 2012 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000

Related Links
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3.3 Damping Modeling (on page 2366)
- V. IBC 2012 Response Spectrum (on page 3422)

TR.32.10.1.10 Response Spectrum Specification per IBC 2015

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2015 edition of the ICC specification International Building Code (IBC) and ASCE 7-10, for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

SPECTRUM comb-method IBC 2015 (TOR) (DECC f20) (ECC f21) *{ X f1
| Y f2 | Z f3 } ACCELERATION

{DAMP f5 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f6) (ZPA f7) ({ DOMINANT f10 | SIGN })
(SAVE) (IMR f11) (STARTCASE f12)

Note: The data from SPECTRUM through ACC must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

{ZIP f8 | LAT f9 LONG f13 | SS f14 S1 f15 } SITE CLASS (f16) (FA f17 FV f18) TL f19

Where:

Table 267: Parameters used for IBC 2015 response spectrum

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DECCENTRICITY f20</td>
<td>1.0</td>
<td>Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).</td>
</tr>
<tr>
<td>ECCENTRICITY f21</td>
<td>0.05</td>
<td>Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.</td>
</tr>
<tr>
<td>Parameter</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td>X ( f_1 ), Y ( f_2 ), Z ( f_3 )</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
</tbody>
</table>
| DAMP \( f_5 \) | 0.05 | The damping ratio. Specify a value of exactly 0.0000011 to ignore damping.  
If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503)  
If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510) |
| MISSING \( f_6 \) | | Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the \( f_6 \) value entered in length/sec\(^2\) (this value is not multiplied by SCALE).  
If \( f_6 \) is zero, then the spectral acceleration at the ZPA \( f_7 \) frequency is used. If \( f_7 \) is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results.  
**Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases. |
| ZPA \( f_7 \) | 33 [Hz] | The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING \( f_6 \) is entered. |
| DOMINANT \( f_{10} \) | 1 (1st Mode) | The dominant mode method. All results will have the same sign as mode number \( f_{10} \) alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).  
**Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode. |
| IMR \( f_{11} \) | 1 | The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through \( f_{11} \) will be used. Missing Mass modes are not output. |
### Parameter | Default Value | Description
--- | --- | ---
STARTCASE \( f_{12} \) | Highest Load Case No. + 1 | The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If \( f_{12} \) is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.
ZIP \( f_{11} \) | | The zip code of the site location to determine the latitude and longitude and consequently the \( S_s \) and \( S_1 \) factors. (IBC 2015, ASCE 7-10 Chapter 22)
LAT \( f_{12} \) | | The latitude of the site used with the longitude to determine the \( S_s \) and \( S_1 \) factors. (IBC 2015, ASCE 7-10 Chapter 22)
LONG \( f_{13} \) | | The longitude of the site used with the latitude to determine the \( S_s \) and \( S_1 \) factors. (IBC 2015, ASCE 7-10 Chapter 22)
SS \( f_{14} \) | Mapped MCE for 0.2s spectral response acceleration. (IBC 2015, ASCE 7-10 Chapter 22) | 
S1 \( f_{15} \) | Mapped spectral acceleration for a 1-second period. (IBC 2000, equation 16-17. IBC 2003, ASCE 7-02 section 9.4.1.2.4-2. IBC 2006, ASCE 7-05 Section 11.4.1) | 
SITE CLASS \( f_{16} \) | Enter A through F for the Site Class as defined in the IBC code. (IBC 2000, Section 1615.1.1 page 350. IBC 2003, Section 1615.1.1 page 322. IBC 2006 ASCE 7-05 Section 20.3) | 
FA \( f_{17} \) | Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3) | 
FV \( f_{18} \) | Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3) | 
TL \( f_{19} \) | Long-Period transition period in seconds. (IBC 2015, ASCE 7-10 Chapter 22) | 

\( \text{comb-method} = \{ \text{SRSS} | \text{ABS} | \text{CQC} | \text{ASCE} | \text{TEN} | \text{CSM} | \text{GRP} \} \) are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

**SRSS** Square Root of Summation of Squares method.

**ABS** Absolute sum. This method is very conservative and represents a worst case combination.
CQC  Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \sqrt{\sum_n \sum_m f^n \rho_{nm} f^m} \]

where

\[ \rho_{nm} = \frac{8 \zeta^2 (1 + r) r^{2/3}}{(1 - r^2)^2 + 4 \zeta^2 r(1 + r)^2} \]

\[ r = \frac{\omega_n}{\omega_m} \leq 1.0 \]

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE  NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.


CSM  Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.


Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

IBC  2015  indicates that the spectrum should be calculated as defined in the IBC 2015 specification.

TORSION  indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See “Torsion” for additional information.

Note: If TORSION is entered on any one spectrum case it will be used for all spectrum cases.

ACCELERATION  indicates that the Acceleration spectra will be entered.

Note: IBC / ASCE 7 does not have provisions for displacement response spectra.

DAMP, MDAMP, and CDAMP  select source of damping input:

- DAMP indicates to use the \( f_2 \) value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification
Select Linear or Logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When FILE filename is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**SIGN**

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

**Caution:** Do not enter DOMINANT parameter with this option.

**SAVE**

This option results in the creation of an acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

**Methodology**

The methodology for calculating the response spectra is defined in ASCE 7-2010, section 11.4. The following is a quick summary:

**a.** Input S_s and S_1 (this could have been searched from database or entered explicitly)

**b.** Calculate

\[ S_{ms} = F_a \times S_s \]

and

\[ S_{m1} = F_v \times S_1 \]

Where:

F_a and F_v are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for F_a and F_v in lieu of table values.

**c.** Calculate

\[ S_{ds} = \frac{2}{3} S_{ms} \]

and

\[ S_{d1} = \frac{2}{3} S_{m1} \]

The spectrum is generated as per section 11.4.5.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688) for additional details on IMR load case generation.

**Steps**

For each mode following steps are performed to include Torsion provision.
1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.

   Thus, design eccentricity \( e_{di} = f20 \times e_{si} + f12 \times b_i \) where \( f20 = 1.0 \) and \( f21 = (\pm) 0.05 \)

   Where:
   
   \( e_{si} \) = static eccentricity between center of mass and center of rigidity at floor \( i \).
   
   \( b_i \) = floor plan dimension orthogonal to the direction of earthquake loading.

3. Torsional moment is calculated at each floor.

   \( M_{ik} = Q_{ik} \times e_{di} \) at floor \( i \) for mode \( k \)

4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.

5. Static analysis of the structure is performed with these nodal forces.

6. The analysis results (i.e., member force, joint displacement, support reaction, etc.) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

Example

Refer to V. IBC 2015 Response Spectrum (on page 3431) for a detailed explanation of this example.

| LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
| JOINT LOAD
| 8 FX 49.035
| 3 6 FX 98.07
| END DEFINE REFERENCE LOADS
| ...
| LOAD 1 LOADTYPE None TITLE RS_X
| SPECTRUM SRSS IBC 2015 X 0.333 ACC DAMP 0.05 LIN
| ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000

Related Links

- V. IBC 2015 Response Spectrum (on page 3431)

TR.32.10.1.11 Response Spectrum Specification per SNiP II-7-81

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per SNiP II-7-81 for dynamic analysis.

General Format

```
SPECTRUM comb-method SNIP A f1 *{ {X f2 | KWX f3 | KX1 f4} | {Y f5 | KWY f6 | KY1 f7} | {Z f8 | KWZ f9 | KZ1 f10} ACCELERATION (SCALE f11)
{DAMP f12 | CDAMP | MDAMP} ( {LINEAR | LOGARITHMIC} ) (MISSING f13) (ZPA f14)
({DOMINANT f15 | SIGN}) SOIL {1 | 2 | 3} (SAVE)
```

The data from SPECTRUM through SCALE above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Where:
### Table 268: Parameters for SNIP II-7-81 response spectrum

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A ( f_1 )</td>
<td></td>
<td>Zoning factor, A, which is based on maximum acceleration factor for the seismic zone. This factor must be modified for SOIL types other than 2. The exact zone factor value used for a specific location requires engineering judgment. The following table serves as a guide for accelerations and corresponding zone factors which would be used.</td>
</tr>
<tr>
<td>X ( f_2 )</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero. Alternatively, you may input individual parameters such as ( KWX ), ( KX1 ) the product of which would be used as the factor along that direction.</td>
</tr>
<tr>
<td>Y ( f_5 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Z ( f_8 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SCALE ( f_{11} )</td>
<td></td>
<td>Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec(^2) units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s(^2) or 32.2 ft/s(^2)).</td>
</tr>
<tr>
<td>DAMP ( f_{12} )</td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503) If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510)</td>
</tr>
<tr>
<td>MISSING ( f_{13} )</td>
<td></td>
<td>Optional parameter to use the “Missing Mass” method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec(^2) for this missing mass mode is the ( f_{13} ) value entered in length per second squared units (this value is not multiplied by SCALE). If ( f_{13} ) is zero, then the spectral acceleration at the ZPA ( f_{14} ) frequency is used. If ( f_{14} ) is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. For SRSS and CQC, the results of this calculation are SRSSed with the modal combination results. If either ( f_{13} ) or ( f_{14} ) are not entered, the defaults will be used. Missing mass does not include the effect of masses lumped at the supports unless the support is a stiff spring or an Enforced support.</td>
</tr>
</tbody>
</table>

**Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ZPA ( f_{14} )</td>
<td></td>
<td>The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING ( f_{13} ) is entered.</td>
</tr>
<tr>
<td>DOMINANT ( f_{15} )</td>
<td></td>
<td>The dominant mode method. All results will have the same sign as mode number ( f_{15} ) alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero).</td>
</tr>
</tbody>
</table>

**Note:** Do not enter the SIGN parameter with this option.

\( \text{comb-method} = \{ \text{SRSS} \ | \text{CQC} \} \) are methods of combining the responses from each mode into a total response.

The CQC method requires damping. The SRSS method does not use damping unless spectra-period curves are made a function of damping (see File option below). CQC includes the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic methods and is recommended.

**SRSS** Square root of summation of squares method as prescribed by the SNiP II-7-81 code.

**CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[
F = \sqrt{\sum_{n} \sum_{m} f_{n} \rho_{nm} f_{m}}
\]

where

\[
\rho_{nm} = \frac{8 \xi^{2}(1 + r) r^{2/3}}{(1 - r^{2})^{2} + 4 \xi^{2} r(1 + r)^{2}}
\]

\[
r = \frac{\omega_{n}}{\omega_{m}} \leq 1.0
\]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.

The specifier SNIP is mandatory to denote that the applied loading is as per the guidelines of SNiP II-7-81.

**ACCELERATION** indicates that the Acceleration spectra will be entered.

**Note:** SNiP II-7-81 / SP 14.13330.2011 do not have provisions for displacement response spectra.

**DAMP, MDAMP, and CDAMP** select source of damping input:

- **DAMP** indicates to use the \( f_{2} \) value for all modes
- **MDAMP** indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
• **CDAMP** indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the **CONSTANT** specification.

**LINEAR or LOGARITHMIC**

Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When **FILE filename** is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**SIGN**

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for **ABS** option.

**Caution:** Do not enter **DOMINANT** parameter with this option.

**SOIL**

Defines the subsoil conditions on which the response spectrum will be generated.

**Note:** The Zoning Factor, \( A \) (\( f_1 \)), must be adjusted for soil types other than type 2.

1. Non-weathered rock and rocklike geological formation or permafrost subsoil.
2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.
3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.

**SAVE**

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec\(^2\). These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

**Description**

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections **TR.32 Loading Specifications** (on page 2650) and **G.17.3 Dynamic Analysis** (on page 2362). An illustration of mass modeling is available, with explanatory comments, in the sample file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Rus\Seismic_Russ.STD.

**Example 1**

The definition of a SNiP response spectrum in the X direction on a structure built on weather rock and where the Zoning Factor is 0.7071. As this is the first load case with a response spectrum, then the masses are modeled as loads.

```
LOAD 2 LOADTYPE Seismic TITLE SPECTRUM IN X-DIRECTION
*Masses
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
```
### Technical Reference of STAAD Commands
#### TR.32 Loading Specifications

<table>
<thead>
<tr>
<th>SELFWEIGHT Z 1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD</td>
</tr>
<tr>
<td>10 FX 17.5</td>
</tr>
<tr>
<td>10 FY 17.5</td>
</tr>
<tr>
<td>10 FZ 17.5</td>
</tr>
<tr>
<td>MEMBER LOADS</td>
</tr>
<tr>
<td>5 CON GX 5.0 6.0</td>
</tr>
<tr>
<td>5 CON GY 5.0 6.0</td>
</tr>
<tr>
<td>5 CON GX 7.5 10.0</td>
</tr>
<tr>
<td>5 CON GY 7.5 10.0</td>
</tr>
<tr>
<td>5 CON GX 5.0 14.0</td>
</tr>
<tr>
<td>5 CON GY 5.0 14.0</td>
</tr>
</tbody>
</table>

*SNiP Spectrum definition
SPECTRUM SRSS SNIP A 0.7071 X 1.0 ACC DAMP 0.05 SCALE 1.0 LIN MIS 0 ZPA 40 SOIL 2

**Note:** The maximum response spectrum load cases allowed in one run is 50.

For full details on Response Spectrum refer to section [TR.32.10.1 Response Spectrum Analysis](#) (on page 2687).

---

### Example 2

STAAD PLANE RESPONSE SPECTRUM ANALYSIS
START JOB INFORMATION
JOB NAME Plane Russian example
ENGINEER DATE 12-Feb-08
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.2 0 0; 3 0 4.5 0; 4 7.2 4.5 0; 5 0 9.0 0; 6 7.2 9.0 0;
MEMBER INCIDENCES
1 1 3; 2 2 4; 3 3 5; 4 4 6; 5 3 4; 6 5 6;
MEMBER PROPERTY RUSSIAN
1 TO 4 PRIS AY 10000 YD 0.6 ZD 0.6
5 TO 6 PRIS YD 0.9 ZD 0.3
SUPPORTS
1 TO 2 FIXED BUT MZ
3 TO 6 FIXED BUT FX
CONSTANTS
E 30.0e+006 ALL
POISSON 0.2 ALL
CUT OFF MODE SHAPE 2
*NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
*WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 1 SEISMIC LOADING

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 4 FX 310.586</td>
</tr>
<tr>
<td>3 4 FY 310.586</td>
</tr>
<tr>
<td>5 6 FX 310.586</td>
</tr>
<tr>
<td>5 6 FY 310.586</td>
</tr>
</tbody>
</table>

SPECTRUM CQC SNIP A 1.0 X 1.0 ACC MIS SOIL 2
PERFORM ANALYSIS
PRINT MODE SHAPES
PRINT STORY DRIFT

---
Technical Reference of STAAD Commands
TR.32 Loading Specifications

Related Links
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3.3 Damping Modeling (on page 2366)

TR.32.10.1.12 Response Spectrum Specification per SP 14.13330.2011

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per SP 14.13330.2011 for dynamic analysis.

General Format

```
SPECTRUM { SRSS | CQC } SP11 (ECCENTRICITY) A f1 *{ { X f2 | Y f3 | Z f4} ACCELERATION
(SCALE f5) (DAMP f6) (LOG) (MIS f7) (ZPA f8) {{ DOMINANT f9 | SIGN }} SOIL { 1 | 2 | 3}
```

The data from SPECTRUM through SCALE above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Where:

Table 269: Parameters for SP 14.13330.2011 response spectrum

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A f1</td>
<td></td>
<td>Zoning factor, A, which is based on maximum acceleration factor for the seismic zone. This factor must be modified for SOIL types other than 2.</td>
</tr>
<tr>
<td>X f2</td>
<td>0.0</td>
<td>Factors for the input spectrum to be applied in X, Y, &amp; Z directions. Any one or all directions can be input. Directions not provided will default to zero.</td>
</tr>
<tr>
<td>Y f3</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>Z f5</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>SCALE f5</td>
<td></td>
<td>A linear scaling factor by which the spectrum data will be multiplied according to the provisions of SP 14.13330.2011.</td>
</tr>
<tr>
<td>DAMP f6</td>
<td>0.05</td>
<td>The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2503) If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2510)</td>
</tr>
</tbody>
</table>


<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MISSING ( f7 )</td>
<td>Optional parameter to use the “Missing Mass” method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec(^2) for this missing mass mode is the ( f7 ) value entered in length per second squared units (this value is not multiplied by SCALE). If ( f7 ) is zero, then the spectral acceleration at the ZPA ( f8 ) frequency is used. If ( f8 ) is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. For SRSS and CQC, the results of this calculation are SRSSed with the modal combination results. If either of ( f7 ) or ( f8 ) are not entered, the defaults will be used. Missing mass does not include the effect of masses lumped at the supports unless the support is a stiff spring or an Enforced support. Note: If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.</td>
<td></td>
</tr>
<tr>
<td>ZPA ( f8 )</td>
<td>33.0 Hz</td>
<td>The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING ( f7 ) is entered.</td>
</tr>
<tr>
<td>DOMINANT ( f9 )</td>
<td>1</td>
<td>The dominant mode method. All results will have the same sign as mode number ( f9 ) alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). Note: Do not enter the SIGN parameter with this option.</td>
</tr>
</tbody>
</table>

\( \text{comb-method = \{ SRSS | CQC \} are methods of combining the responses from each mode into a total response.} \)

The CQC method requires damping. The SRSS method does not use damping unless spectra-period curves are made a function of damping (see File option below). CQC includes the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic methods and is recommended.

**SRSS** Square root of summation of squares method as prescribed by the SNiP II-7-81 code.

**CQC** Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

\[ F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m} \]

where

\[ \rho_{nm} = \frac{8\zeta^2(1+r)^2}{(1-r^2)^2 + 4\zeta^2r(1+r)^2} \]
\[ r = \frac{\omega_n}{\omega_m} \leq 1.0 \]

**Note:** The cross-modal coefficient array is symmetric and all terms are positive.

The specifier **SP11** is mandatory to denote that the applied loading is as per the guidelines of SP 14.13330.2011.

**ECCENTRICITY** automatic introduction of artificial mass eccentricity according to SP 14.13330.2011.

**ACCELERATION** indicates that the Acceleration spectra will be entered.

**Note:** SNiP II-7-81 / SP 14.13330.2011 do not have provisions for displacement response spectra.

**DAMP, MDAMP, and CDAMP** select source of damping input:
- **DAMP** indicates to use the \( f_2 \) value for all modes
- **MDAMP** indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

**LINEAR or LOGARITHMIC**
Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When **FILE filename** is entered, the interpolation along the damping axis will be linear.

**Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

**SIGN** This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

**Caution:** Do not enter DOMINANT parameter with this option.

**SOIL** Defines the subsoil conditions on which the response spectrum will be generated.

**Note:** The Zoning Factor, \( A (f_1) \), must be adjusted for soil types other than type 2.

1. Non-weathered rock and rocklike geological formation or permafrost subsoil.
2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.
3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.

**Description**
Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement.
For more information on dynamic mass modeling, refer to sections TR.32 Loading Specifications (on page 2650) and G.17.3 Dynamic Analysis (on page 2362)

```
Example

STAAD PLANE RESPONSE SPECTRUM ANALYSIS
START JOB INFORMATION
JOB NAME Plane Russian example
ENGINEER DATE 12-Feb-08
END JOB INFORMATION
UNIT METER KN

JOINT COORDINATES
1 0 0 0; 2 7.2 0 0; 3 0 4.5 0; 4 7.2 4.5 0; 5 0 9.0 0; 6 7.2 9.0 0;
MEMBER INCIDENCES
1 1 3; 2 2 4; 3 3 5; 4 4 6; 5 3 4; 6 5 6;
MEMBER PROPERTY RUSSIAN
1 TO 4 PRIS AY 10000 YD 0.6 ZD 0.6
5 TO 6 PRIS YD 0.9 ZD 0.3
SUPPORTS
1 TO 2 FIXED BUT MZ
3 TO 6 FIXED BUT FX
CONSTANTS
E 30.0e+006 ALL
POISSON 0.2 ALL
CUT OFF MODE SHAPE 1
*NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
*WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 1 SEISMIC LOADING
JOINT LOAD
3 4 FX 310.586
3 4 FY 310.586
5 6 FX 310.586
5 6 FY 310.586
SPECTRUM CQC SP11 ECC A 1.0 X 1.0 ACC MIS SOIL 2
PERFORM ANALYSIS
PRINT MODE SHAPES
PRINT STORY DRIFT
PRINT ANALYSIS RESULTS
FINISH
```

Related Links

- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3.3 Damping Modeling (on page 2366)

**TR.32.10.2 Time Varying Load for Response History Analysis**

Used to apply loads which are defined with a changing magnitude of acceleration, force, or moment (refer to TR.31.4 Definition of Time History Load (on page 2630)). These can either be assigned to specific nodes (as a FORCE or MOMENT type) and/or globally to the all the supports on the model as a ground motion (as an ACCELERATION type). In addition to the data produced from a general modal analysis, a load case which includes time history loading will produce a set of graphs in the post processing mode to indicate how the displacement, velocity or acceleration of a selected node changes in each of the three global directions over the time period of the applied dynamic loading.
General Format

The following set of commands may be used to model time history loading on the structure for response time history analysis. Nodal time histories and ground motion time histories may both be provided within one load case.

**TIME LOAD**

| joint-List *{ FX | FY | FZ | MX | MY | MZ } | It | Ia | f2 |

**GROUND MOTION** { ABSOLUTE | (RELATIVE ) } { X | Y | Z } | It | Ia | f2

Parameter | Description
--- | ---
ABSOLUTE | nodal results are absolute (elastic response + motion of ground). If entered on any ground command, all results will be absolute.
RELATIVE | nodal results are relative (elastic response). (Default if neither ABS or REL are specified)
It | sequential position in the input data of type number of time varying load. To refer to first type number entered, use a 1 here regardless of actual type number entered. Ground Motion must have an Acceleration Type; Time Load forces must have a Force type; and Time Load moments must have a Moment Type (refer to TR.31.4 Definition of Time History Load on page 2630).
Ia | Arrival time number (integer). This is the sequential number of the arrival time in the list explained in TR.31.4 Definition of Time History Load on page 2630. Thus the arrival time number of a3 is 3 and of a_n is n.
f2 | The Force, Moment, or Acceleration Amplitude at this joint and direction will be multiplied by this factor (default = 1.0). For accelerations, if the amplitude-time curve was in g's, please use the Scale Factor in the Define Time History command to convert g's to the acceleration units used in that command. This is recommended due to possible unit changes between that command and this command.

**Note:** Multiple loads at a joint-direction pair for a particular (It Ia) pair will be summed. However there can be no more than four (It Ia) pairs associated with a particular joint-direction pair, the first four such entries will be used. Loads at slave joint directions will be ignored.

Either **TIME LOAD** or **GROUND MOTION**, or both may be specified under one load case. More than one load case for time history analysis is not permitted.

For **TIME LOAD** data, multiple direction specifiers can be in one entry as follows (the direction specifiers must be on one line and missing values are assumed to be 1):

**TIME LOAD**

| It | Ia | joint-List *
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>3</td>
<td>FX</td>
<td>1</td>
<td>FZ</td>
<td>1</td>
<td>4</td>
<td>-2.1</td>
</tr>
<tr>
<td>6</td>
<td>7</td>
<td>FX</td>
<td>FY</td>
<td>FZ</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

"STAAD.Pro" 2768 User Manual
Notes

a. A Time History analysis requires the mode shapes. These are calculated using the mass matrix determined from the loading specified in the first dynamic load case. (refer to TR.34.2 Modal Calculation Command (on page 2791))

b. The Node Displacement table reports the maximum displacement that occurs at each node over the entire time range.

c. The displacement of the model at a specific time instance can be displayed by using the time slider bar on the Results toolbar when displaying a load case with time history loading.

d. If any Node Groups are defined, the time history graphs can be set to display the average results for the group. The name of the selected node or group being displayed is given in the graph title bar.

e. A model can include only one load case with time history loads.

Note: STAAD.Pro is also capable of generating floor spectrum responses for a time history analysis. Refer to TR.37.10 Floor Spectrum Command (on page 2833) for details on adding this to the analysis commands.

Example

LOAD 1
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
MEMBER LOADS
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GZ 7.5 10.0
TIME LOAD
2 3 FX 1 3
5 7 FX 1 6
GROUND MOTION REL X 2 1

In this example, the permanent masses in the structure are provided in the form of selfweight and member loads (refer to TR.32 Loading Specifications (on page 2650) and G.17.3 Dynamic Analysis (on page 2362)) for obtaining the mode shapes and frequencies. The mass model can also be created using a Reference Load case with the Load type of Mass (refer to TR.31.6 Defining Reference Load Types (on page 2642)). The rest of the data is the input for application of the time varying loads on the structure. Forcing function type 1 is applied at joints 2 and 3 starting at arrival time number 3. (Arrival time number 3 is 1.8 seconds in example shown in TR.31.4 Definition of Time History Load (on page 2630)). Similarly, forcing function type 1 is applied at joints 5 and 7 starting at arrival time number 6 (4.4 seconds). A ground motion (type 2) acts on the structure in the x-direction starting at arrival time number 1 (0.0 seconds).

Related Links

- G.17.3.5 Response Time History (on page 2373)
- M. To add a time history load (on page 861)
- G.17.3 Dynamic Analysis (on page 2362)
- G.17.3.2 Mass Modeling (on page 2365)
- G.17.3.5 Response Time History (on page 2373)
TR.32.11 Repeat Load Specification

This command is used to create a primary load case using combinations of previously defined primary load cases.

General Format

```
REPEAT LOAD
i1, f1, i2, f2 ... in, fn
```

Where:

- \( i1, i2 ... in \) : primary load case numbers
- \( f1, f2 ... fn \) : corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. Limit of 550 prior cases may be factored. Prior cases to be factored may also contain the REPEAT LOAD command.

Description

This command may be used to create a primary load case using combinations of previously defined primary load case(s). The REPEAT LOAD differs from the load COMBINATION command (See TR.35 Load Combination Specification (on page 2791)) in two ways:

1. A REPEAT LOAD is treated as a new primary load. Therefore, a P-Delta analysis will reflect correct secondary effects. (LOAD COMBINATIONS, on the other hand, algebraically combine the results such as displacements, member forces, reactions and stresses of previously defined primary loadings evaluated independently).
2. In addition to previously defined primary loads, you can also add new loading conditions within a load case in which the REPEAT LOAD is used.
3. The REPEAT LOAD option is available to factor prior load case data and add those forces into the current load case.

The load case data types that will be factored include Joint Loads, Member loads, Element loads, Inertia Loads, Fixed End loads, Selfweight loads, Displacements, Area loads, Prestress loading, and Temperature loads. Floor loads, Wind loads Snow loads, and UBC loads are first converted to equivalent member loads and then factored.

Modal dynamic analysis load cases (Response Spectrum, Time History, Steady State) should not be used in REPEAT LOAD. It is also not available for loads generated using some of the program's load generation facilities such as MOVING LOAD Generation. However load cases with WIND LOAD may be used in Repeat Load.

UBC cases may only be used in REPEAT LOAD if there is a PERFORM ANALYSIS and CHANGE command after each UBC case. See notes with UBC LOAD command.

Prestress on a given member from 2 or more load cases cannot be combined.

Example:
```
LOAD  1 DL + LL
SELFWEIGHT Y -1.4
MEMBER LOAD
  1 TO 7 UNIFORM Y -3.5
LOAD  2 DL + LL + WL
```
4. For a load case that is defined using the `REPEAT LOAD` attribute, the constituent load cases themselves can also be `REPEAT LOAD` cases. See load case 4 below.

```
LOAD 1
  SELFWEIGHT Y -1.0
LOAD 2
  MEMBER LOAD
  2 UNI GY -1.5
LOAD 3
  REPEAT LOAD
      1 1.5
LOAD 4
  REPEAT LOAD
      2 1.2 3 1.25
```

**Related Links**

- *M. To add a repeat load case* (on page 872)
- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2352)

**TR.32.12 Generation of Loads**

This command is used to generate Moving Loads, UBC Seismic loads and Wind Loads using previously specified load definitions.

Primary load cases may be generated using previously defined load systems. The following sections describe generation of moving loads, UBC seismic loads and Wind Loads.

**TR.32.12.1 Generation of Moving Loads**

This command is used to generate Moving Loads using previously specified load definitions.

Predefined moving load system types may be used to generate the desired number of primary load cases, each representing a particular position of the moving load system on the structure. This procedure will simulate the movement of a vehicle in a specified direction on a specified plane on the structure.

**General Format**

```
LOAD GENERATION n (ADD LOAD i )
  TYPE j x1 y1 z1 *( XINC f1 | YINC f2 | ZINC f3 ) ( { YRANGE | ZRANGE } r )
```

Where:

- **n** total number of primary load cases to be generated
- **i** load case number for the previously defined load case to be added to the generated loads
- **j** type number of previously defined load system
- **x1, y1, z1** x, y and z coordinates (global) of the initial position of the reference load
- **f1, f2, f3** x, y, or z (global) increments of position of load system to be used for generation of subsequent load cases.
Use only XINC & ZINC if Y up; Use only XINC & YINC if Z up.

\( r \) (Optional) defines section of the structure along global vertical direction to carry moving load. This \( r \) value is added and subtracted to the reference vertical coordinate (\( y_1 \) or \( z_1 \)) in the global vertical direction to form a range. The moving load will be externally distributed among all members within the vertical range thus generated. \( r \) always should be a positive number. In other words, the program always looks for members lying in the range \( y_1 \) and \( y_1 + \text{abs}(r) \) or \( z_1 \) and \( z_1 + \text{abs}(r) \). The default \( r \) value is very small, so entering \( r \) is recommended.

The ADD LOAD specification may be used to add a previously defined load case to all the load cases generated by the LOAD GENERATION command. In the example below, the SELFWEIGHT specified in load case 1 is added to all the generated load cases.

Sequential load case numbers will be assigned to the series of generated primary load cases. Numbering will begin at one plus the immediate previous load case number. Allow for these when specifying load cases after load case generation.

Notes

a. Primary load cases can be generated from Moving Load systems for frame members only. This feature does not work on finite elements.

b. This facility works best when the roadway, as well as the movement of the vehicle are along one of the global horizontal (X or Z) or (X or Y) directions.

For bridge decks which are skewed with respect to the global axes, the load generation may not yield the most satisfactory results. In such cases, the M. Bridge Deck workflow (on page 883) is recommended. The Bridge Deck workflow works on the influence line/influence surface method, and is considerably superior to the moving load generator. It also has the advantage of being able to calculate the critical load positions on decks modeled using plate elements.

c. The \( x_1, y_1, z_1 \) values of the starting position of the reference wheel must be provided bearing in mind that the reference wheel has to be at the elevation of the deck. An improper set of values of these parameters may result in the wheels being positioned incorrectly, and consequently, no load may be generated at all.

Example

```
LOAD 1 DL Only
SELFWEIGHT
LOAD GENERATION 20 ADD LOAD 1
  TYPE 1 0. 5. 10. Xi 10.
  TYPE 2 0. 10. 10. Zi 15.
LOAD 22 Live Load on Pavement
MEMBER LOAD
  10 TO 20 30 TO 40 UNI GY -5.0
LOAD COMBINATION 31 10 0.75 22 0.75
PERFORM ANALYSIS
```

Related Links

- M. To add vehicles to the load generation (on page 855)
- G.16.1 Moving Load Generator (on page 2343)
- M. To generate moving load cases (on page 854)
- G.16.1 Moving Load Generator (on page 2343)
TR.32.12.2 Generation of Seismic Loads

This command is used to generate UBC Seismic loads using previously specified load definitions.

Related Links
- G.16.2 Seismic Load Generator (on page 2343)
- G.16.2 Seismic Load Generator (on page 2343)
- G.16.2 Seismic Load Generator (on page 2343)
- G.16.2 Seismic Load Generator (on page 2343)
- G.16.2 Seismic Load Generator (on page 2343)
- G.16.2 Seismic Load Generator (on page 2343)
- G.16.2 Seismic Load Generator (on page 2343)
- G.16.2 Seismic Load Generator (on page 2343)

TR.32.12.2.1 Generation of Seismic Loads

Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications. The following general format should be used to generate loads in a particular direction.

**General Format**

```
LOAD i
```

```
code LOAD { X | Y | Z } (f1) (DECCENTRICITY f2) (ACCIDENTAL f3)
```

```
code = { UBC | IBC | 1893 | AIJ | COL | CFE | NTC | RPA }
```

Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD i</td>
<td>-</td>
<td>Load case number.</td>
</tr>
<tr>
<td>X</td>
<td>Y</td>
<td>Z f1</td>
</tr>
<tr>
<td>DEC f2</td>
<td>0.0</td>
<td>Multiplying factor for natural torsion –arising due to static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm– to be used to multiply the seismic horizontal torsion load. Must be a positive value (greater than 1.0) or exactly 0.0.</td>
</tr>
</tbody>
</table>
Parameter | Default Value | Description
--- | --- | ---
**ACC** | 1.0 | Multiplying factor for accidental torsion, to be used to multiply the seismic accidental torsion load. May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Use only horizontal directions.

To include horizontal torsional moment arising due to static eccentricity for a rigid floor diaphragm, the following conditions must be satisfied:

a. The floor must be modeled as a rigid diaphragm.
b. A positive value (greater than 1.0) for DEC must be provided. Seismic load is applied at center of mass instead of center of rigidity which incorporates the effect that a value less than or equal to 1.0 will yield. Placing seismic load at center of mass of a rigid diaphragm automatically includes inherent torsion in analysis corresponding to static eccentricity (the difference between center of mass and center of rigidity). Providing DEC parameter as 0.0 for a model having rigid diaphragm to ignore inherent torsion is not possible.
c. The ACC command must not be present in seismic definition (i.e., in the DEFINE code LOAD command). If present, the natural torsion factor will be ignored and only the accidental torsion for all seismic loads will be considered.

The design eccentricity for calculating horizontal torsion is the DEC + ACC values. When ACC is negative, it becomes DEC - ACC (i.e., the torsion magnitudes are always additive).

**Dynamic Eccentricity**

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

In most country’s seismic codes, the provision for design eccentricity \(e_{di}\) at \(i^{th}\) floor of a building is given by the following equations:

\[
\begin{align*}
e_{di} &= \alpha e_{si} + \beta b_i \\
e_{di} &= \delta e_{si} - \beta b_i
\end{align*}
\]

where

\[
\begin{align*}
e_{si} &= \text{static eccentricity at } i^{th} \text{ floor} \\
b_i &= \text{plan dimension of the } i^{th} \text{ floor normal to the direction of ground motion} \\
\alpha, \beta, \text{ and } \delta &= \text{specified constants}
\end{align*}
\]

If \(\alpha = 1.0, \delta = 1.0, \text{ and } \beta = 0.05\) parameters to be defined are DEC = 1.0 and ACC = 0.05 in one load case and DEC = 1.0 and ACC = -0.05 in another load case to include both natural and accidental torsion.

Only TOR ECC 0.05 or TOR ECC -0.05 can also be defined without specifying DEC 1.0 since it is the default that is included in the analysis.
The cases (d) and (e) are followed i.e. seismic load is passed through CM instead of CR. Natural torsion is automatically included in analysis for DEC ≤ 1.0 i.e., no additional inherent torsion is applied. However, if DEC > 1.0, a twisting moment with modified eccentricity of DEC-1 will act at CM. In this case, a message occurs in the output.

**Example**

Define UBC load
Zone 0.2 K 1.0 I 1.5 TS 0.5
Selfweight
Joint Weight
1 to 100 Weight 5.0
101 to 200 Weight 7.5
Load 1 UBC in X-Direction
UBC Load X DEC 1.0 ACC 0.05
Joint Load
5 25 30 FY -17.5
Perform Analysis
Change
Load 2 UBC in X-Direction
UBC Load X DEC 1.0 ACC -0.05
Joint Load
5 25 30 FY -17.5
Perform Analysis
Change
Load 3 UBC in Z-Direction
UBC Load Z DEC 0.0 ACC 0.05
Perform Analysis
Change
In the above example, notice that the first three load cases are UBC load cases. They are specified before any other load cases.

**Notes**

**a.** The UBC load cases should be provided as the first set of load cases. Non-UBC primary load case specified before a UBC load case is not acceptable. Additional loads such as MEMBER LOADS and JOINT LOADS may be specified along with the UBC load under the same load case.

Example of **Incorrect** Usage: The error here is that the UBC cases appear as the 3rd and 4th cases, when they should be the 1st and 2nd cases.

Example of Correct Usage

**b.** If the UBC cases are to be factored later in a Repeat Load command; or if the UBC case is to be used in a tension/compression analysis; or if Re-analysis (two analysis commands without a CHANGE or new load case in between); then each UBC case should be followed by PERFORM ANALYSIS then CHANGE commands as
shown in the example above. Otherwise the PERFORM ANALYSIS then CHANGE can be omitted. Using the CHANGE command will require the SET NL command to define the maximum number of load cases being entered in the analysis. Also LOAD LIST ALL should be entered after the last PERFORM ANALYSIS command.

Example of Incorrect Usage: The error here is that the CHANGE command is missing before Load Case 2.

```
Load 1
UBC Load X 1.2
Selfweight Y -1
Joint Load
  3 FY -4.5
PDELTA Analysis
Load 2
UBC Load Z 1.2
Selfweight Y -1
Joint Load
  3 FY -4.5
PDELTA Analysis
```

Example of Correct Usage

```
Load 1
UBC Load X 1.2
Selfweight Y -1
Joint Load
  3 FY -4.5
PDELTA Analysis
Change
Load 2
UBC Load Z 1.2
Selfweight Y -1
Joint Load
  3 FY -4.5
PDELTA Analysis
Change
```

c. Up to 8 UBC cases may be entered.

d. The REPEAT LOAD specification cannot be used for load cases involving UBC load generation unless each UBC case is followed by an analysis command then CHANGE.

Example

```
Load 1
UBC Load X 1.0
PDELTA Analysis
Change
Load 2
Selfweight Y -1
PDELTA Analysis
Change
Load 3
Repeat Load
  1 1.4 2 1.2
PDELTA Analysis
```

e. If UBC load generation is performed for the X and the Z (or Y if Z up) directions, the command for the X direction must precede the command for the Z (or Y if Z up) direction.

Related Links
- [G.16.2 Seismic Load Generator](on page 2343)
M. To add a seismic load (on page 842)

TR.32.12.2.2 Generation of IS:1893 Seismic Load

The following general format should be used to generate the IS 1893 load in a particular direction.

**General Format**

```
LOAD i

1893 LOAD { X | Y | Z } (f1) (DECCENTRICITY f2) (ACCIDENTAL f3)
```

Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD i</td>
<td></td>
<td>Load case number.</td>
</tr>
<tr>
<td>X</td>
<td>Y</td>
<td>Z f1</td>
</tr>
<tr>
<td>DEC f2</td>
<td>0.0</td>
<td>Multiplying factor for natural torsion – arising due to static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm – to be used to multiply the seismic horizontal torsion load. Must be a positive value (greater than 1.0) or exactly 0.0.</td>
</tr>
<tr>
<td>ACC f3</td>
<td>1.0</td>
<td>Multiplying factor for accidental torsion, to be used to multiply the seismic accidental torsion load. May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).</td>
</tr>
</tbody>
</table>

Use only horizontal directions.

**Note:** Refer to “IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load” (on page 2579) for additional information on using this command in conjunction with IS 1893 static seismic loads.

**Example**

In the above example, the first two load cases are the 1893 load cases. They are specified before any other load case.

```
Define 1893 Load
Zone 0.05 RF 1.0 I 1.5 SS 1.0
Selfweight
Joint Weight
7 to 12 Weight 17.5
13 to 20 Weight 18.0
Member Weight
1 to 20 UNI 2.0
Load 1 1893 Load in X-Direction
1893 Load X
Joint Load
5 25 30 FY -17.5
Load 2 1893 Load in Z-Direction
1893 Load Z
Load 3 Dead Load
Selfweight
```
TR.32.12.3 Generation of Wind Loads

This command is used to generate Wind Loads using previously specified load definitions.

The built-in wind load generation facility can be used to calculate the wind loads based on the parameters defined in TR.31.3 Definition of Wind Load (on page 2623). The following general format should be used to perform the wind load generation. See G.16.3 Wind Load Generator (on page 2345) for the two types of structures on which the load can be generated. For closed type structures, the vertical panel areas bounded by beam members only (and ground), and exposed to the wind, are used to define loaded areas (plates and solids are ignored). The loads generated are applied only at the joints at vertices of the bounded areas. For open type structures also, generation is done considering only the members in the model.

The automated load generator should only be used for vertical panels. Panels not parallel to the global Y axis (for Y UP) should be loaded separately.

General Format

```
LOAD i
WIND LOAD (-){ X | Y | Z } (f) TYPE j (OPEN) { XR f1 f2 | YR f1 f2 | ZR f1 f2 | LIST memb-list | ALL }
```

Where:

**LOAD i**  
load case number

{-}{X | Y | Z}  
Direction of wind in global axis system. Use horizontal directions only.

**f**  
The factor to be used to multiply the wind loads. Negative signs may be used to indicate opposite direction of resulting load (default=1.0).

**TYPE j**  
Type number of previously defined systems

**OPEN**  
optional word to be used if loading is to be generated on "open" type of structures. If this not specified, load will be generated assuming the panels are “closed”.

**XR f1 f2**  
global coordinate values to specify X or Y or Z range for member selection

Using X, -X, Z or –Z and the f factor. With respect to the axis, a minus sign indicates that suction occurs on the other side of the selected structure. If all of the members are selected and X (or Z) is used and the factor is positive, then the exposed surfaces facing in the –x (or –z) direction will be loaded in the positive x (or z) direction (normal wind in positive direction). See diagrams that follow. If X and a negative factor is used, then the exposed surfaces facing in the +x direction will be loaded in the negative x direction (normal wind in negative direction). [If –X is entered and a negative factor, then the exposed surfaces facing in the -x direction will be loaded in the negative x direction (suction). If –X is entered and a positive factor, then the exposed surfaces facing in the +x direction will be loaded in the positive x direction (suction).]
A member list or a range of coordinate values (in global system) may be used. All members which have both end coordinates within the range are assumed to be candidates (for closed type structures) for defining a surface which may be loaded if the surface is exposed to the wind. The loading will be in the form of joint loads (not member loads). 1, 2, or 3 ranges can be entered to form a “layer”, “tube”, or “box” for selecting members in the combined ranges. Use ranges to speed up the calculations on larger models.

Example

```
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.12 HEIGHT 100 200
EXP 0.6 JOI 1 TO 25 BY 7 29 TO 37 BY 4 22 23
TYPE 2
INT 0.1 0.12 HEIGHT 100 900
EXP 0.3 YR 0 500
LOAD 1
SELF Y -1.0
LOAD 2
WIND LOAD Z 1.2 TYPE 2 ZR 10 11
LOAD 3
WIND LOAD X TYPE 1 XR 7 8 ZR 14 16
```
Example
For open structures
LOAD 1 WIND LOAD IN Z DIRECTION
WIND LOAD 2 -1.2 TYPE 1 OPEN

Notes

a. For closed type structures, panels or closed surfaces are generated by the program based on the members in the ranges specified and their end joints. The area within each closed surface is determined and the share of this area (influence area) for each node in the list is then calculated. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

Hence, one should make sure that the members/joints that are exposed to the wind make up a closed surface (ground may form an edge of the closed surface). Without a proper closed surface, the area calculated for the region may be indeterminate and the joint force values may be erroneous. Consequently, the number of exposed joints should be at least three.

b. Plates and solids are not considered for wind load generation. On such entities, wind must be applied using pressure loading facilities for plates and solids.

General Format for Russian SP or SNiP Wind Loads
The format for applying wind loads per the Russian codes are as follows:

```
LOAD i
WIND LOAD { X | Z | -X | -Z} f1 ( OBLIQUE ) CONFIGURATION k ( NU fNU )
TYPE j { XRANGE f1 f2 | YRANGE f1 f2 | ZRANGE f1 f2 | LIST member-list | ALL }
```
The value X or Z defines the wind direction. Use of a negative sign defines a leeward wind load.

Where:

\[ f \] the wind pressure factor. A negative value defines a reverse wind loading.

**OBLIQUE** An option that indicates the wind direction is 45 degrees to the X or Z axes and is *only* used with a SNiP 1985 wind definition.

\[ k \] a configuration parameter defined as follows:

- **Prismatic buildings:**
  
  For a SNiP 1985 wind load definition, the valid range of values is -1 to 12, where these values represent:
  
  -1. Prismatic building structure - rectangular building. Outstanding architectural details on the left facade.
  0. Prismatic building structure - rectangular building. Both facade faces are smooth (no outstanding architectural details).
  1. Prismatic building structure - rectangular building. Outstanding architectural details on the right facade.
  2. Prismatic building structure - rectangular building. Outstanding architectural details on both side facades.
  4. Prismatic building structure - rhombic building.
  5. to 12. Prismatic building structure - number of polygon vertices in building from 5 to 12.

  For a SNiP 2011 wind load definition, the valid range of values is either 0, or 5 to 12, where these values represent:

  0. Prismatic building structure - rectangular building. Both facade faces are smooth (no outstanding architectural details).
  (1 to 4 are not used)
  5. to 12. Prismatic building structure - number of polygon vertices in building from 5 to 12.

- **For 'stick' structures,** where the structure is modeled as a set of vertical members. This type is typically used to define a cylindrical stack or chimney structure. Thus, the model does not define a closed panel such as defined by a frame formed from a number of columns and beams. In this case, the configuration parameter is instead used to define the surface roughness according to SP 20.13330.2011 table D.11, where the parameter range is 0 to 10 as follows:

  **Table 270: Parameter if values for stick-type, cylindrical structures**

<table>
<thead>
<tr>
<th>k</th>
<th>Surfacing Material</th>
<th>Roughness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Ideal surface</td>
<td>0.000</td>
</tr>
<tr>
<td>1</td>
<td>Glass</td>
<td>0.0015</td>
</tr>
<tr>
<td>2</td>
<td>Polished metal</td>
<td>0.002</td>
</tr>
<tr>
<td>3</td>
<td>Fine paint</td>
<td>0.006</td>
</tr>
<tr>
<td>k</td>
<td>Surfacing Material</td>
<td>Roughness (mm)</td>
</tr>
<tr>
<td>----</td>
<td>----------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>4</td>
<td>Spray paint</td>
<td>0.02</td>
</tr>
<tr>
<td>5</td>
<td>Cast iron</td>
<td>0.2</td>
</tr>
<tr>
<td>6</td>
<td>Galvanized steel</td>
<td>0.2</td>
</tr>
<tr>
<td>7</td>
<td>Smooth concrete</td>
<td>0.2</td>
</tr>
<tr>
<td>8</td>
<td>Rough concrete</td>
<td>1.0</td>
</tr>
<tr>
<td>9</td>
<td>Rust</td>
<td>2.0</td>
</tr>
<tr>
<td>10</td>
<td>Brickwork</td>
<td>3.0</td>
</tr>
</tbody>
</table>

\( f_{NU} \)  the wind pressure correlation coefficient. If the parameter is 1, a computed value is used. For rectangular buildings, the calculated value is always used (any input is ignored).

**Note:** The Russian wind load routines establish the applied wind forces from a dynamic response of the structure. Either a dynamic load case should be defined before any wind load cases or loads that represent masses should also be included in the first load case that includes Russian wind load assignments. This in effect becomes a dynamic load case. Thus, if the effects of these forces are to be combined to the effects of static load cases, then this should be done using a LOAD COMBINATION command.

**General Format for SP 20.13330.2016 Wind Loads**

STAAD.Pro can generate both static and dynamic wind loads per the SP 20.13330.2016 code.

```
LOAD i

(mass-data)

WIND LOAD { X | Z | -X | -Z } f1 (OBLIQUE) WALONG f2 WACROSS f3 (FRQ) TYPE j (DYN k)

TYPE j { XRANGE f1 f2 | YRANGE f1 f2 | ZRANGE f1 f2 | LIST member-list | ALL }
```

Where:

- **mass-data** Optional mass load defined within the dynamic wind load when a separate load case for modal analysis is not previously defined in the STAAD input file.
- **f1** Correction factor along specified direction. This needs to be used in combination with the WIND LOAD setting.
- **OBLIQUE** Optional parameter that indicates the wind direction is 45 degrees to the X and Z axes and is with a SNiP 2016 wind definition.
WALONG $f_2$  Effective length of the structure parallel to wind direction.

WACROSS $f_3$  Effective projection of the structure facing the wind direction.

FRQ  Optional parameter which results in computation of dynamic wind load vector for all mode shapes extracted from Modal Analysis provided it is within code stipulated frequency limit. Absence of the parameter will result in calculation taking the 1st mode shape only.

TYPE $j$  Wind definition type number. This definition must be of type SP20 2016. Both static and dynamic wind loads per the 2016 Russian code use this wind definition.

DYN $k$  Optional parameter that indicates that the wind loading should include the dynamic component determined from the modal response and the static wind load defined in primary load case $k$. If this option is omitted, then only the static loads from the load definition will be calculated.

Dynamic Wind Load per Russian SP 20.13330.2016

In order to perform dynamic wind load generation in STAAD.Pro you must previously define a static load case. The static load case could be any primary static load case which is defined before the Russian dynamic load case, including a static wind loading per the same code. This static load case will provide static load vector to the dynamic wind load module.

As the Russian dynamic wind load component requires modal masses and eigen vectors to calculate the dynamic wind load component at nodes, modal analysis must be performed before the dynamic wind load definition. Therefore, you must also include a separate load case for modal analysis with reference mass defined before the load case. See G.17.3.2 Mass Modeling (on page 2365) for details. Alternatively, if the mass loads are not needed for use with other load cases (that is, a reference load case is not needed for other loads), then you may define the mass loads within the dynamic load case prior to the WIND LOAD command.

The overall wind load effect will be determined by the sum of effects of the static load and an SRSS of all of the contributing modes of the dynamic wind effect which will be used to increase the magnitude of the static load effect.

$$ w = w_m + w_p $$

where

\[ w_m = \text{mean wind pressure} \]
\[ w_p = \text{quasi-static wind fluctuation component} \]

STAAD.Pro uses the static wind load and the modal results to create a combined dynamic load from which modal loads are calculated. The modal loads are analyzed and the overall dynamic effect is determined by combining all these with an SRSS combination which is used to increase the effect of the static wind component.

Related Links
TR.32.13 Generation of Snow Loads

This command is used to generate Snow Loads using previously specified Snow load definitions. This input should be a part of a load case.

General Format

```
SNOW LOAD
_flr_group TYPE j CS f1 { BALANCED | UNBALANCED } { OBSTRUCTED | UNOBSTRUCTED } { MONO | HIP | GABLE }
```

Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>_flr_group</td>
<td>-</td>
<td>The members that form the roof and that are to be loaded by snow load must be listed in a floor group (See TR.16 Entities as Single Objects (on page 2440)).</td>
</tr>
<tr>
<td>TYPE j</td>
<td>-</td>
<td>The type number of previously defined snow load system.</td>
</tr>
<tr>
<td>CS f1</td>
<td>0.0</td>
<td>Roof Slope factor (CS). For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.</td>
</tr>
<tr>
<td>BALA or UNBA</td>
<td>BALA</td>
<td>Balanced or unbalanced snow load. Default is balanced. These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.</td>
</tr>
<tr>
<td>OBST or UNOB</td>
<td>UNOB</td>
<td>Obstructed or unobstructed.</td>
</tr>
<tr>
<td>MONO or HIP or GABLE</td>
<td>NONO</td>
<td>Roof geometry type.</td>
</tr>
</tbody>
</table>

Use as many floor groups and types as necessary in each load case.

Related Links

- **G.16.4 Snow Load** (on page 2345)
- **M. To add an ASCE 7-02 snow load** (on page 851)
- **G.16.4 Snow Load** (on page 2345)

TR.32.14 Notional Loads

A notional load is a lateral load (horizontal load) which is derived from an existing vertical load case. This load type has been introduced to accommodate a requirement in design codes. The AISC 360-05 specification for
example defines notional loads as lateral loads that are applied at each framing level and are specified in terms of gravity loads.

**Description**

Both Principal and Reference load cases can be selected and moved into the Notional Load Definition where the required factor and direction can be specified. The notional loads are calculated and applied as joint loads.

**Note:** The actual values of the applied loading are not displayed in the User Interface until after the analysis has been performed.

**General Format**

LOAD nLoad LOADTYPE type TITLE title

... 

Load items

... 

NOTIONAL LOAD

n \{ X | Z \} (f1)

Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>n</td>
<td>-</td>
<td>Load case number of the primary load case or reference load case which contains the vertical load items.</td>
</tr>
<tr>
<td>f1</td>
<td>0.002 (or user-specified)</td>
<td>Factor by which the contents of LN are to be multiplied. Typically, codes recommend 0.2 to 0.3 % (0.002 to 0.003). The default is 0.002 or the factor specified in DIRECT ANALYSIS definition block.</td>
</tr>
</tbody>
</table>

Multiple load items can be specified under any NOTIONAL LOAD and there can be one or more blocks of NOTIONAL LOAD command(s) in any load case.

**Note:** Notional Loads can be derived from gravity load cases specified under the DEFINE REFERENCE LOAD table which are described in TR.31.6 Defining Reference Load Types (on page 2642).

**Example 1**

LOAD 1 : DEAD
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -100
LOAD 2: DEAD NOTIONAL LOAD
NOTIONAL LOAD
1 X 0.002
... 
LOAD 10: IMPOSED
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -50
Example 2

If we want to combine the load cases in Example 1, two methods are available:

LOAD 3 : DEAD LOAD + IMPOSED LOAD + Notional Loads
REPEAT LOAD
1 1.0 10 1.0
NOTIONAL LOAD
1 X 0.002 10 X 0.002

As an alternative, the following could be used:

LOAD 3 : DEAD LOAD + IMPOSED LOAD + Notional Loads
REPEAT LOAD
1 1.0 2 1.0 10 1.0 11 1.0

Example 3

Similarly,

Load 1 LOADTYPE None TITLE DEAD
SELFWEIGHT Y -1.15
MEMBER LOAD
143 145 CON GY -16

Load 2 LOADTYPE None TITLE DEAD-NOTIONAL
NOTIONAL LOAD
1 X 1.0

Load 3 LOADTYPE None TITLE DEAD AND NOTIONAL
REPEAT LOAD
1 1.0
NOTIONAL LOAD
1 X 1.0

Alternatively the following could be used to define load case 3:

Load 3 LOADTYPE None TITLE DEAD AND NOTIONAL
REPEAT LOAD
1 1.0 2 1.0

Example 4

Using Notional loads with REFERENCE loads

DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead TITLE REF DEAD LOAD
SELFWEIGHT Y -1 LIST 1 TO 120
FLOOR LOAD
YRANGE 19 21 FLOAD -0.05 GY
YRANGE 39 41 FLOAD -0.05 GY
LOAD R2 LOADTYPE Live TITLE REF LIVE LOAD
FLOOR LOAD
YRANGE 19 21 FLOAD -0.03 GY
YRANGE 39 41 FLOAD -0.03 GY
END DEFINE REFERENCE LOADS

**********************************************
* LOAD COMBINATIONS INCLUDING NOTIONAL LOADS *
**********************************************

*DEAD + LIVE +/- NOTIONAL LOAD
*
LOAD 1
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 X 0.0024 R2 X 0.0032
LOAD 2
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 X -0.0024 R2 X -0.0032
LOAD 3
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 Z 0.0024 R2 Z 0.0032
LOAD 4
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 Z -0.0024 R2 Z -0.0032

If a PRINT STATIC CHECK or PRINT LOAD DATA command is added with any analysis specification in the input file, the generated NOTIONAL LOADs would be printed as in the following:

<table>
<thead>
<tr>
<th>LOAD</th>
<th>JOINT</th>
<th>DIRECTION</th>
<th>LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>6117</td>
<td>X</td>
<td>-0.00200 X</td>
<td>74124.266 = 148.249</td>
</tr>
<tr>
<td>6118</td>
<td>X</td>
<td>-0.00200 X</td>
<td>115504.569 = 231.009</td>
</tr>
<tr>
<td>6119</td>
<td>X</td>
<td>-0.00200 X</td>
<td>38397.157 = 76.794</td>
</tr>
<tr>
<td>6120</td>
<td>X</td>
<td>-0.00200 X</td>
<td>15551.155 = 31.102</td>
</tr>
<tr>
<td>6121</td>
<td>X</td>
<td>-0.00200 X</td>
<td>29341.723 = 58.695</td>
</tr>
<tr>
<td>6122</td>
<td>X</td>
<td>-0.00200 X</td>
<td>34006.254 = 68.013</td>
</tr>
<tr>
<td>6123</td>
<td>X</td>
<td>-0.00200 X</td>
<td>39771.147 = 79.542</td>
</tr>
<tr>
<td>6124</td>
<td>X</td>
<td>-0.00200 X</td>
<td>39712.750 = 79.426</td>
</tr>
<tr>
<td>6125</td>
<td>X</td>
<td>-0.00200 X</td>
<td>34006.254 = 68.013</td>
</tr>
</tbody>
</table>

48554642.872 -97109.290

Related Links
- G.17.2.1.4 AISC 360 Direct Analysis (on page 2352)
- M. To add a notional load case (on page 852)
- G.17.2.1.4 AISC 360 Direct Analysis (on page 2352)
TR.33 Reference Load Cases - Application

Describes how you can call the data specified under those types in actual load cases. Reference Load types are described in TR.31.6 Defining Reference Load Types (on page 2642).

General Format

The format of a reference to a Reference Load in a primary load (j) case is thus:

\[
\text{LOAD } j \text{ LOADTYPE (type) load_title REFERENCE LOAD R(i) 1.0} \ldots
\]

Example

\[
\begin{align*}
\text{LOAD 1 LOADTYPE None TITLE D+L} \\
\text{REFERENCE LOAD} \\
R1 & 1.0 R2 1.0 \\
\text{LOAD 2 LOADTYPE None TITLE DEAD+SNOW} \\
\text{REFERENCE LOAD} \\
R1 & 1.0 R3 1.0 \\
\text{LOAD 3 LOADTYPE None TITLE D+H} \\
\text{REFERENCE LOAD} \\
R1 & 1.0 R4 1.0 \\
\text{ELEMENT LOAD} \\
1212 & 1267 TRAP GY JT -0.54 -0.44 -0.44 -0.54.
\end{align*}
\]

Related Links

- G.17.2.1.4 AISC 360 Direct Analysis (on page 2352)

TR.34 Frequency Calculation

There are two methods available in STAAD for calculating the frequencies of a structure:

1. an approximate method called the Rayleigh method and
2. a more exact method which involves the solution of an eigenvalue problem.

Both methods are explained in the following sections.

Related Links

- G.17.3 Dynamic Analysis (on page 2362)
- G.17.3 Dynamic Analysis (on page 2362)
- G.17.3 Dynamic Analysis (on page 2362)
- G.17.3.1 Solution of the Eigenproblem (on page 2363)
- G.17.3 Dynamic Analysis (on page 2362)
TR.34.1 Rayleigh Frequency Calculation

This command may be used to calculate the Rayleigh method approximate frequency of the structure for vibration corresponding to the general direction of deflection generated by the load case that precedes this command. Thus, this command typically follows a load case.

General Format

```
CALCULATE RAYLEIGH (FREQUENCY)
```

Description

This command is specified after all other load specifications of any primary load case for which the Rayleigh frequency is calculated. This Rayleigh frequency calculation is based on the Rayleigh iteration method using 1 iteration. If a more accurate, full-scale eigensolution is required, the MODAL CALCULATION command (see next section) may be used. A full eigensolution is automatically performed if a RESPONSE SPECTRUM or TIME HISTORY is specified in any load case.

Example

```
LOADING 1
SELFWEIGHT X 1.0
CALCULATE RAYLEIGH FREQUENCY
LOADING 2
SELFWEIGHT Z 1.0
CALCULATE RAYLEIGH FREQUENCY
LOADING 3 WIND LOAD
```

In this example, the Rayleigh frequency is calculated for the X direction mode of vibration in load case 1, and the Z direction in case 2.

In case 1, the structure is being displaced by SELFWEIGHT applied statically along global X. For most frames which are supported only at their base, this produces a deflected shape resembling the lowest mode shape along global X. Hence, this frequency will resemble that for the lowest X direction mode.

Similarly, the frequency calculated for load case 2 ought to be similar to that for the lowest Z direction mode because the selfweight is applied statically along global Z.

**Tip:** If you want a mode or frequency to be for lateral motion (X or Z), then enter the loads in the X or Z directions. Entering loads in the vertical direction when the lowest modes are in the lateral direction is a common mistake.

The output will consist of the value of the frequency in cycles per second (cps), the maximum deflection along with that global direction and the joint number where that maximum occurs.

Notes

This command is based on the Rayleigh method of iteration using 1 iteration. The frequency calculated estimates the frequency as if the structure were constrained to vibrate in the static deflected shape generated by the loads in the load case.
In many instances, the forces should be in one global direction to get the mode and frequency associated with that direction.

**Related Links**
- M. To calculate the structure frequency (on page 834)
- G.17.3 Dynamic Analysis (on page 2362)

**TR.34.2 Modal Calculation Command**

This command may be used to obtain a full scale eigensolution to calculate relevant frequencies and mode shapes. It should not be entered if this case or any other case is a TIME LOAD or RESPONSE SPECTRUM case. For Steady State/Harmonic analysis this command must be included in the load case that defines the weights and weight moment of inertias for eigensolutions.

**General Format**

```
MODAL CALCULATION ( REQUESTED ) ( MISSING MASS )
```

This command is typically used in a load case after all loads are specified. The loads will be treated as weights and weight moment of inertias for eigensolutions (see G.17.3 Dynamic Analysis (on page 2362) and TR.32 Loading Specifications (on page 2650)). You are advised to specify the loads keeping this in mind.

This case will be independently solved statically and dynamically. Static results using the loads will include joint displacements, member forces, support reactions, and other outputs computed from a normal static analysis without any dynamic effects included.

STAAD.Pro can include the stress stiffening effect (geometric stiffness) based on the axial member forces/plate in-plane stresses from a selected load case when calculating the modes & frequencies of a structure.

In addition, the Dynamic results (using loads as masses) will include Mode Shapes and Frequencies.

**Note:** The MODAL CALCULATION command can be included in any of the primary load cases, but only in one of them.

For Steady State/Harmonic analysis, enter the MISSING MASS parameter to include the missing mass procedure in the analysis.

**Related Links**
- M. To calculate the structure frequency (on page 834)
- G.17.3 Dynamic Analysis (on page 2362)
- G.17.3 Dynamic Analysis (on page 2362)
- EX. US-28 Calculation of Modes and Frequencies of a Bridge (on page 4591)
- EX. UK-28 Calculation of Modes and Frequencies of a Bridge (on page 4873)
- M. To calculate the structure frequency (on page 834)

**TR.35 Load Combination Specification**

This command may be used to combine the results of the analysis. The structure will not be analyzed for the combined loading. The combination of results may be algebraic, SRSS, a combination of both algebraic and SRSS, or Absolute.
Note: The LOAD COMBINATION specification is not appropriate for obtaining secondary effects for combined loads. Refer to notes below.

General Format

LOAD COMBINATION ( { SRSS | ABS } ) ( GENERATE ) i a1
i1 f1 i2 f2 ... fSRSS

Where:

i Load combination number. This can be any integer smaller than 100,000 that is not the same as any previously defined primary load case or load combination number.

a1 Any title for the load combination.

i1, i2, ... represents the load case or load combination numbers which are to be combined.

f1, f2, ... represents corresponding factors to be applied to loadings. A value of 0.0 is also allowed.

fSRSS optional factor to be applied as a multiplying factor on the combined result of the SRSS load combination (see examples below).

GENERATE Optional command for SRSS or ABS load combinations which will generate load combinations representing all possible positive or negative actions in each load degree of freedom. When used, a single set of absolute value analysis results is replaced by these 64 load combination analysis results.

A hyphen may be used at the end of a line to continue this command onto the next line.

Description

If the load combination option is left out (i.e., neither ABS or SRSS is included), then the results from analyses will be combined algebraically.

LOAD COMBINATION 6 DL+LL+WL
  1 0.75 2 0.75 3 1.33

If the ABS load combination method is included, then the absolute value of results from the analyses will be combined.

Note: If GEN is included this single set of absolute values is replaced by 64 sets of load combinations.

LOAD COMBINATION ABS 7 DL+LL+WL
  1 0.85 2 0.65 3 2.12

If the SRSS load combination method is included, then the results from analyses may be combined both algebraically and using the SRSS (Square Root of Summation of Squares) method. The combination scheme may be mixed if required. For example, in the same load combination case, results from load cases may be combined in the SRSS manner and then combined algebraically with other load cases.

Note: The case factor is not squared.

Note: If GEN is included this single set of absolute values is replaced by 64 sets of load combinations.
Generate Load Combinations

If the optional GENERATE command is included, this single set of absolute values is replaced by 64 sets of load combinations. Otherwise, the program will follow the usual method of ABS or SRSS load combination.

For a particular model, all 13 section forces for each of the members for the internally generated 64 load combination cases can be printed in an external text file. To do this, use the following SET command in the input file before joint incidences are specified.

```
SET CG TXT
```

This command will instruct the program to print the section forces for each member for 1 to 64 generated load combinations to a file named \textit{filename}_CG.txt located in the same location as the STAAD input file.

**Caution:** For models with significantly larger number of members or load combination cases generated, this file can become extremely large. This output option should only be used when the individual cases require review and not in normal circumstance.

Response Spectra in Load Combinations

The following steps are followed in order to derive member section forces for a response spectrum case.

\textbf{a.} Forces at start and end joints of a member are calculated for each mode.

\textbf{b.} Since masses are lumped at two member ends, it is assumed that the section forces due each of the modes are varying linearly between member ends. Thus at any section the section force is linearly interpolated for each mode.

\textbf{c.} The forces at each section for all modes are combined using modal combination method to arrive at the final result.

The section forces for each mode for any response spectrum load case can be printed in an external text file. To do this, use the following SET command in the input file before joint incidences are specified.

```
SET RS TXT
```

This command will instruct the program to print the section forces for each member to a file named \textit{filename}_RESP.txt located in the same location as the STAAD input file.

**Caution:** For models with significantly larger number of members or load combination cases generated, this file can become extremely large. This output option should only be used when the individual cases require review and not in normal circumstance.

Once the section forces are available it can be combined with other static load cases using SRSS/ABS and GENERATE options.

Design for Load Combinations per ASME NF

Steel design is performed for active load cases that may include primary loads and load combinations. If any load combination is an internal load generation case, instead of designing for one particular load combination case, the design is performed for all 64 load combinations. Thus, the steel design results may vary from what if it had been designed for only one combination case. If any load combination generation case becomes a critical load case for design, it is noted as such in the design output.
All section forces for the 64 generated load cases—which parent load combination case becomes the critical design load—can be printed in an external text file. To do this, use the following SET command in the input file before joint incidences are specified.

```
SET NF TXT
```

This command will instruct the program to print the section forces for each member to a file named `filename_RESP.txt` located in the same location as the STAAD input file.

**Caution:** For models with significantly larger number of members or load combination cases generated, this file can be extremely large. This output option should only be used when the individual cases require review and not used in normal circumstance.

---

**Example of Simple SRSS Combination**

Several combination examples are provided to illustrate the possible combination schemes

```
LOAD COMBINATION SRSS 8 DL+SEISMIC
1 1.0 2 0.4 3 0.4
```

This (LOAD COMBINATION SRSS 8) illustrates a pure SRSS load combination with a default SRSS factor of one. The following combination scheme will be used:

\[
v = 1.0 \cdot (1 \cdot L1^2 + 0.4 \cdot L2^2 + 0.4 \cdot L3^2)^{1/2}
\]

where

\[
v \quad = \quad \text{the combined value}
\]

\[
L1, L2, \text{and } L3 \quad = \quad \text{values from load cases 1, 2 and 3.}
\]

Since an SRSS factor is not provided, the default value of 1.0 is being used.

---

**Notes**

a. In the **LOAD COMBINATION SRSS** option, if the minus sign precedes any load case no., then that load case will be combined algebraically with the SRSS combination of the rest.

b. If the secondary effects of combined load cases are to be obtained through a P-Delta, Member/Spring Tension/Compression, Multilinear Spring, or Nonlinear Analysis, then the **LOAD COMBINATION** command is inappropriate for the purpose. See the **REPEAT LOAD** command (TR.32.11 Repeat Load Specification (on page 2770)) for details.

c. In a load combination specification, a value of 0 (zero) as a load factor is permitted. In other words, a specification such as

```
LOAD COMB 7
1 1.35 2 0.0 3 1.2 4 0.0 5 1.7
```

is permitted. This is the same as

```
LOAD COMB 7
1 1.35 3 1.2 5 1.7
```

d. All combination load cases must be provided immediately after the last primary load case.

e. The total number of primary and combination load cases combined cannot exceed the limit described in TR.2 Problem Initiation and Model Title (on page 2409).

f. A maximum of 550 load cases can be combined using a **LOAD COMBINATION** command.
g. Load combinations can refer to previously defined load combination numbers.

**Note:** This feature requires build 20.07.07.30 or higher.

### Examples of Algebraic & SRSS Combination in the Same Load Combination Case

**Example 1**

<table>
<thead>
<tr>
<th>LOAD COMBINATION</th>
<th>SRSS</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1</td>
<td>0.75</td>
<td>2</td>
</tr>
<tr>
<td>1.3</td>
<td>3</td>
<td>2.42</td>
</tr>
</tbody>
</table>

The combination formula will be as follows:

\[
v = 0.75 \cdot L_1 + 0.75 \cdot (1.3 \cdot L_2^2 + 2.42 \cdot L_3^2)^{1/2}
\]

where

- \( v \) = the combined value
- \( L_2 \) and \( L_3 \) = values from load cases 2 and 3

In the above specification, a minus sign precedes load case 1. Thus, Load 1 is combined algebraically with the result obtained from combining load cases 2 and 3 in the SRSS manner. The SRSS factor of 0.75 is applied on the SRSS combination of 2 and 3.

**Example 2**

<table>
<thead>
<tr>
<th>LOAD COMBINATION</th>
<th>SRSS</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1</td>
<td>0.75</td>
<td>-2</td>
</tr>
<tr>
<td>0.572</td>
<td>3</td>
<td>1.2</td>
</tr>
<tr>
<td>1.7</td>
<td>0.63</td>
<td></td>
</tr>
</tbody>
</table>

Here, both load cases 1 and 2 are combined algebraically with the SRSS combination of load cases 3 and 4. Note the SRSS factor of 0.63. The combination formula will be as follows:

\[
v = 0.75 \cdot L_1 + 0.572 \cdot L_2 + 0.63 \cdot (1.2 \cdot L_3^2 + 1.7 \cdot L_4^2)^{1/2}
\]

**Related Links**

- [M. To define a new load combination](on page 866)
- [M. To automatically generate load combinations](on page 869)

---

### TR.36 Calculation of Problem Statistics

This item has been removed. Please contact the Bentley Technical Support Group for further details.

### TR.37 Analysis Specification

STAAD analysis options include linear static analysis, P-Delta (or second order analysis), Nonlinear analysis, and several types of Dynamic analysis.

This command is used to specify the analysis request. In addition, this command may be used to request that various analysis related data, like load info, statics check info, etc. be printed.

**Related Links**

- [G.17.1 Stiffness Analysis](on page 2346)
TR.37.1 Linear Elastic Analysis

Used to perform a static, linear elastic analysis on the structure.

General Format

```
PERFORM ANALYSIS (PRINT { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL })
```

Without one of these analysis commands, no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

The following PRINT options are available:

- If the PRINT LOAD DATA command is specified, the program will print an interpretation of all the load data.
- PRINT STATICS CHECK will provide a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
- PRINT STATICS LOAD prints everything that PRINT STATICS CHECK does, plus it prints a summation of all internal and external forces at each joint. This option generates a large volume of output.

**Note:** Since PRINT STATICS LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

- PRINT BOTH is equivalent to PRINT LOAD DATA plus PRINT STATICS CHECK.
- PRINT ALL is equivalent to PRINT LOAD DATA plus PRINT STATICS LOAD.
- The PRINT MODE SHAPES command may be added separately after the Analysis command if mode shapes are desired. Refer to TR.42 Print Specifications (on page 2840) for using additional print specifications.

Notes

This command directs the program to perform the analysis that includes:

a. Checking whether all information is provided for the analysis;
b. Forming the joint stiffness matrix;
c. Checking the stability of the structure;
d. Solving simultaneous equations, and
e. Computing the member forces and displacements.
f. If a RESPONSE SPECTRUM, TIME LOAD, or GROUND MOTION is specified within a load case or the MODAL CALCULATION command is used, a dynamic analysis is performed.

**Note:** Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.

General Analysis Comments

STAAD.Pro allows multiple analyses in the same run. Multiple analyses may be used for the following purposes:

a. Successive analysis and design cycles in the same run result in optimized design. STAAD.Pro automatically updates changes in member cross-sectional sizes. Thus the entire process is automated.

Refer to Example 1 in the Getting Started & Examples manual for detailed illustration.
Multiple analyses may be used for load-dependent structures. For example, structures with bracing members are analyzed in several steps. The bracing members are assumed to take Tension load only. Thus, they need to be activated and inactivated based on the direction of lateral loading.

The entire process can be modeled in one STAAD.Pro run using multiple PERFORM ANALYSIS commands. STAAD.Pro is capable of performing a design based on the load combinations provided. Refer to Example 4 in the Getting Started & Examples manual for detailed illustration.

c. You may also use Multiple Analyses to model change in other characteristics like supports, releases, section properties, etc.

d. Multiple Analyses may require use of additional commands like the SET NL command (TR.5 Set Command Specification (on page 2413)) and the CHANGE command.

e. Analysis and CHANGE are required after UBC cases if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

Related Links
- A. To specify a linear elastic analysis (on page 928)
- G.17 Analysis Facilities (on page 2346)

TR.37.2 P-Delta Analysis Options

Used to perform a second-order analysis, considering either large- or small-delta effects—or both—on the structure.

General Format

The following options are available in STAAD.Pro when performing a P-Delta analysis:

**Note:** Options 1 and 2 in the present form are effective from STAAD.Pro 2007 Build 06.

1. P-Delta analysis with Small & Large Delta effects or Large Delta effects only. With this option the global forces are adjusted with every iteration

   \[ \text{PDELTA} \,(n) \,\text{ANALYSIS} \,(\text{CONVERGE} \,(m)) \,(\{ \text{LARGEDELTA} \,|\, \text{SMALLDELTA} \}) \,(\text{PRINT} \,\text{print-options}) \]

   Where:
   - \( n \) = no. of iterations desired (default value of \( n = 1 \)).
   - \( m \) = the maximum number of iterations used to check convergence, even if convergence has not been achieved. Do not use CONVERGE when \( n \) is specified.
   - SMALLDELTA is the default

2. P-Delta analysis including stress stiffening effect of the KG matrix. With this option the global stiffness is adjusted with every iteration

   \[ \text{PDELTA} \,\text{KG} \,\text{ANALYSIS} \,(\text{PRINT} \,\text{print-options}) \]

   \text{print-options} = \{ \text{LOAD \,DATA} \,|\, \text{STATICS \,CHECK} \,|\, \text{STATICS \,LOAD} \,|\, \text{BOTH} \,|\, \text{ALL} \}

   See TR.37.1 Linear Elastic Analysis (on page 2796) for details.

   Without one of these P-Delta analysis commands, no P-Delta analysis will be performed.

   These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.
A PDELTA ANALYSIS will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (TR.32.11 Repeat Load Specification (on page 2770)). Secondary effects will not be evaluated correctly for load combinations.

P-Delta effects are computed for frame members and plate elements only. They are not calculated for solid elements or curved beams.

**Notes for Small or Large Delta (Option 1)**

a. This command directs the program to perform the analysis that includes:
   a. Checking whether all information is provided for the analysis;
   b. Forming the joint stiffness matrix
   c. Checking the stability of the structure;
   d. Solving simultaneous equations, and
   e. Computing the member forces and displacements.
   f. For P-Delta analysis, forces and displacements are recalculated, taking into consideration the chosen P-Delta effect.
   g. If a RESPONSE SPECTRUM, TIME LOAD, or GROUND MOTION is specified within a load case or the MODAL CALCULATION command is used, a dynamic analysis is performed.

   **Note:** Computing P-Delta effects for dynamic load cases is not recommended since such effects are not considered.

h. In each of the iterations of the PDELTA ANALYSIS, the load vector will be modified to include the secondary effect generated by the displacements caused by the previous iterations.

b. The default procedure of option 1 is based on “P-small & large Delta” effects.(sometimes referred to as P-δ & P-Δ). Enter the LargeDelta parameter to only include the “PlargeDelta” effects (P-Δ only). SmallDelta is recommended.

c. This PDELTA (n) ANALYSIS command (option 1) should specify 3 to 30 iterations to properly incorporate the P-Delta effect. With this many iterations, the PDELTA (n) ANALYSIS SMALLDELTA command results are as good as or better than the PDELTA KG (option 2) command results for static analysis. The advantage of this PDELTA (n) ANALYSIS command comes from not having to re-form and then triangular factorize the stiffness matrix for every iteration within every case. Also this command allows tension/compression.

d. Be aware that global buckling can occur in P-Delta analysis, resulting in large or infinite or NaN values for displacement. Do not use the results of such a case. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for P-Delta analyses.

e. When the CONVERGE command is not specified, the member end forces are evaluated by iterating n times (once, if n is not specified).

   **Caution:** Do not enter n when CONVERGE is provided.

f. When the CONVERGE command is included, the member end forces are evaluated by performing a convergence check on the joint displacements. In each step, the displacements are compared with those of the previous iteration in order to check whether convergence is attained based on the convergence displacement tolerance. In case m is specified, the analysis will stop after that iteration even if convergence has not been achieved. If convergence is achieved in less than m iterations, the analysis is terminated.

   **Note:** The convergence option is not recommended.

To set the convergence displacement tolerance, enter one of the following:
• **SET DISPLACEMENT i_2** command, where \( i_2 \) = a target displacement value in current units. The default value is the maximum span of the structure divided by 120.

• **SET PDELTA TOL i_9** command, where \( i_9 \) = a displacement threshold for convergence in current units of length. The default value is 0.01 inch. If the maximum change in displacement from two consecutive iterations is less than \( i_{tol} \), then that load case is converged.

Refer to **TR.5 Set Command Specification** (on page 2413) for additional information.

**Note:** Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.

**Example**

Following are some examples on use of the command for P-Delta analysis as described in option 1 (on page 2797).

```
PDELTA ANALYSIS
PDELTA 5 ANALYSIS
PDELTA ANALYSIS CONVERGE
PDELTA ANALYSIS CONVERGE 5
PDELTA 20 ANALYSIS SMALLDELTA PRINT STATICS CHECK
```

STAAD.Pro allows multiple P-Delta analyses in the same run (see the General Comments section of **TR.37.1 Linear Elastic Analysis** (on page 2796) for details).

**Notes for Stress Stiffening Matrix (Option 2)**

The P-Delta analysis also provides the option of including the stress stiffening effect of the Kg matrix into the member / plate stiffness.

**a.** A regular STAAD P-Delta Analysis (option 1) performs a first order linear analysis and obtains a set of joint forces from member/plates based on both large and small P-Delta effects. In contrast, the P-Delta KG Analysis (that is, with the Kg option selected) includes the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is then performed using the original load vector. Large & small P-Delta effects are always included in this KG option.

**b.** This command directs the program to perform the analysis that includes:

**a.** Solving the static case.

**b.** Reforming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces.

**c.** Solving simultaneous equations for displacements;

**d.** If a RESPONSE SPECTRUM, TIME LOAD, or GROUND MOTION is specified within a load case or the MODAL CALCULATION command is used, a dynamic analysis is performed. The static cases solved for a PDELTA KG analysis command will be solved first then the dynamic analysis cases.

**Note:** The stiffness matrix used in the dynamic analysis will be the K+Kg matrix used in the last iteration for the last static case. This is a stress stiffened dynamic analysis, sometimes known as a PDELTA Dynamic analysis.

**c.** A **PDELTA KG ANALYSIS** will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (**TR.32.11 Repeat Load Specification** (on page 2770)). Secondary effects will not be evaluated correctly for load combinations since only final results are combined.
d. P-Delta KG effects are computed for frame members and plate elements only. They are not calculated for solid elements. The results are based on "P-large & small Delta" effects.

e. For static analysis, the other P-Delta command \([ \text{PDELTA} (n) \text{ ANALYSIS (SMALL)}]\) with 20 or more iterations (option 1) is preferred.

f. Tension/compression only members are *not* allowed with this PDELTA KG command. You must use the PDELTA (n) ANALYSIS (SMALL) command instead.

g. Be aware that global buckling can occur in a PDELTA KG ANALYSIS. This condition is usually detected by STAAD.Pro. A message is issued (in the basic solver negative L-matrix diagonals may be reported) and the results for that case are set to zero. STAAD.Pro will continue with the next load case.

h. Global buckling may not be detected which could result in a solution with large or infinite or NaN values for displacement or negative L-matrix diagonals or stability errors. Do not use the results of such cases. This condition may require a nonlinear analysis. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for P-Delta analyses.

**Example**

PDELTA KG ANALYSIS PRINT BOTH
PDELTA KG 2 ANALYSIS

**Related Links**

- [G.17.2.1 P-Delta Analysis](on page 2349)
- [A. To specify a P-Delta analysis](on page 928)
- [G.17.2.1.1 P-Delta Analysis – Large Delta and Small Delta](on page 2349)
- [G.17.2.1.2 P-Delta Kg Analysis](on page 2350)
- [G.17.2.1.3 P-Delta K+Kg Dynamic Analysis](on page 2351)

**TR.37.3 Nonlinear Cable Analysis**

**General Format**

For basic cable analysis:

PERFORM CABLE ANALYSIS BASIC (STEPS \(f_1\)) (EQITERATIONS \(f_2\)) (EQTOLERANCE \(f_3\)) (SAGMINIMUM \(f_4\)) (STABILITY \(f_5\) \(f_6\)) (KSMALL \(f_7\)) (PRINT print-options)

**Note:** Use of the Basic cable analysis requires including the BASIC option in the command.

For advanced cable analysis:

PERFORM CABLE ANALYSIS (ADVANCED) (STEPS \(f_8\)) (EQITERATIONS \(f_9\)) (EQTOLERANCE \(f_{10}\)) (REFORM \(f_{11}\)) (KGEOM \(f_{12}\)) (PRINT print-options)

**Note:** Use of the Advanced Cable analysis feature requires the Advanced Analysis License. The ADVANCED option is the default if you have the license.

Where:

print-options = \{ LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL \}

See [TR.37.1 Linear Elastic Analysis](on page 2796) for details.
See [TR.42 Print Specifications](on page 2840) for details on including the cable sag in the output.

This command may be continued to the next line by ending with a hyphen.

### Table 271: Parameters for basic cable analysis

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>STEPS $f_1$</td>
<td>145</td>
<td>The number of load steps for basic cable analysis. The applied loads will be applied gradually in this many steps. Each step will be iterated to convergence. If entered, the value should be in the range 5 to 145.</td>
</tr>
<tr>
<td>EQITERATIONS $f_2$</td>
<td>300</td>
<td>Maximum number of iterations permitted in each load step. Should be in the range of 10 to 500.</td>
</tr>
<tr>
<td>EQTOLERANCE $f_3$</td>
<td>0.0001</td>
<td>The convergence tolerance for the above iterations.</td>
</tr>
<tr>
<td>SAGMINIMUM $f_4$</td>
<td>0.0</td>
<td>Cables (not trusses) may sag when tension is low. This is accounted for by reducing the E value. Sag minimum may be between 1.0 (no sag E reduction) and 0.0 (full sag E reduction).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>As soon as SAGMIN becomes less than 0.95, the possibility exists that a converged solution will not be achieved without increasing the steps to 145 or the pretension loads. The Eq iterations may need to be 300 or more. The Eq tolerance may need to be greater or smaller.</td>
</tr>
<tr>
<td>STABILITY $f_5$</td>
<td>1.0</td>
<td>A stiffness matrix value ($f_5$) that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for the first $f_6$ load steps. The amount added linearly decreases with each of the $f_6$ load steps. If $f_5$ is entered, use 0.0 to 1000.0. This parameter alters the stiffness of the structure.</td>
</tr>
</tbody>
</table>
### Parameter Details

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( f_6 )</td>
<td>1</td>
<td>The number of load steps over which ( f_5 ) is gradually applied. The default is one step.</td>
</tr>
<tr>
<td>KSMALL ( f_7 )</td>
<td>0.0</td>
<td>A stiffness matrix value that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for every load step. The range for ( f_7 ) is between 0.0 and 1.0. This parameter alters the stiffness of the structure.</td>
</tr>
<tr>
<td>STEPS ( f_8 )</td>
<td>1</td>
<td>Used to divide the loading into small increments. Any positive integer is valid. Generally, a value larger than one (1) may be beneficial for solution convergence, because the loading is applied gradually. However, with cable elements having fixed prestressing, a single step typical converges faster.</td>
</tr>
<tr>
<td>EQITERATIONS ( f_9 )</td>
<td>300</td>
<td>Maximum number of iterations permitted in each load step. During each loading increment, the analysis will iterate to find the converging solution. Once the iteration number reaches the maximum value, the analysis will be stopped even though converged solution is not achieved. Any positive integer is valid. Values too small may prevent the solution to achieve convergence. However, excessively large values may cost unnecessary running time when the problem diverges.</td>
</tr>
</tbody>
</table>

**Table 272: Parameters for advanced cable analysis**
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EQTOLERANCE</td>
<td>1.0E-6</td>
<td>The convergence tolerance for the iterations specified in EQITERATIONS. The residual force norm is used for this convergence. This error threshold indicates when the nonlinear solver will stop iterating and consider the ongoing step as converged once the computed error is equal or smaller than this value. A value that is too small may prevent the solution to be considered as converged. However, value that is too larger may result in inaccurate results. The tolerance printed in the output file is the norm of the current out-of-balance forces, divided by the norm of all the externally applied forces. This calculated tolerance must be less than f10 for the current iteration to converge.</td>
</tr>
<tr>
<td>REFORM</td>
<td>1</td>
<td>Used to specify if full Newton-Raphson method or modified Newton-Raphson method will be used. Theoretically, the full Newton-Raphson method can approach converge with less iterations but may require more computation effort than modified Newton-Raphson method.</td>
</tr>
<tr>
<td>KGEOM</td>
<td>0</td>
<td>Used to specify whether or not geometric stiffness will be used.</td>
</tr>
</tbody>
</table>

**Notes**

a. STAAD allows multiple analysis in the same run (see the General Comments in TR.37.1 Linear Elastic Analysis (on page 2796)).

b. Multiple analyses may require use of additional commands like the SET NL command and the CHANGE command.

c. Analysis and CHANGE are required between primary cases for PERFORM CABLE ANALYSIS BASIC or PERFORM CABLE ANALYSIS ADVANCED.

d. A cable element cannot be loaded with either a non-uniformly distributed load or a concentrated load (using a MEMBER LOAD command).
Note: Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.

Related Links

- [A. To specify a nonlinear cable analysis](on page 931)
- [G.17.2.7 Nonlinear Cable or Truss Analysis](on page 2357)
- [G.8.2.1 Linearized Cable Members](on page 2331)
- [G.8.2.2 Nonlinear Cable and Truss Members](on page 2332)
- [G.17.2.7 Nonlinear Cable or Truss Analysis](on page 2357)
- [G.8.2.3 Nonlinear Cable Members for Advanced Cable Analysis](on page 2332)
- [G.17.2.8 Advanced Nonlinear Cable Analysis](on page 2358)

TR.37.4 Buckling Analysis

General Format

PERFORM BUCKLING ANALYSIS (EIGEN) (MAXSTEPS f1) (PRINT print-options)

print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL }

See [TR.37.1 Linear Elastic Analysis](on page 2796) for details.

Without this command (or any of the analysis commands described earlier), no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

Where:

- **EIGEN** if specified the analysis will perform the eigen method, otherwise the iterative method will be used
- **MAXSTEPS f1** the maximum no. of iterations desired (default value of n = 20). 15 is recommended. For use with the iterative (ANALYSIS) method only.

There are two procedures available:

- i. the iterative method: PERFORM BUCKLING ANALYSIS
- ii. the eigen method: PERFORM BUCKLING ANALYSIS EIGEN

For the basic solver, only the iterative method may be used. Both options are available for the advanced solver.

Note: Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.

Iterative Method

This command directs the program to perform an analysis that includes:

- a. Solving the static case.
- b. Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces and inplane plate stresses.
c. Solving simultaneous equations for displacements;
d. Repeat b) and c) for the number of required additional iterations; either until convergence or until MAXSTEPS is reached.

If the loads must be in the opposite direction, STAAD.Pro will stop solving that case at 1 iteration. The results for the case will be outputted; then STAAD.Pro will continue with the next case.

Convergence occurs when two consecutive buckling factors in the iteration are within 0.1% of each other. Results are based on the highest successful Buckling Factor estimate that was calculated; as if the original applied loads times the buckling factor had been entered.

Example with Basic Solver:
```
PERFORM BUCKLING ANALYSIS MAXSTEPS 15 -
PRINT LOAD DATA
```

**Eigen Method**

**Note:** MAXSTEPS input is ignored when using the eigen method.

This command directs the program to perform the analysis that includes:

a. Solving the static case.
b. Reforming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces & inplane plate stresses..
c. Solving an eigenvalue problem for up to 4 buckling factors and buckling shapes

A Buckling Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification ([TR.32.11 Repeat Load Specification](on page 2770)). Buckling will not be performed for LOAD COMBINATIONS cases.

Buckling Kg matrices are computed for frame members and plate elements only. They are not calculated for solid elements or curved members. The results are based on “P-large & small Delta” effects (refer to Option 1 in [TR.37.2 P-Delta Analysis Options](on page 2797)).

Buckling Analysis solves for Buckling Factors. These are the amounts by which the load case must be factored for the buckling shape to occur. Only the last buckling case will be presented in the post processing. However, the buckling factors of all buckling cases will be written to the output.

If the loads must be in the opposite direction, STAAD.Pro will compute negative buckling factors.

Results are for the normalized buckling shape not as if the original applied loads times the buckling factor had been entered.

Example with eigen method:
```
PERFORM BUCKLING ANALYSIS EIGEN
```

**Related Links**
- [G.17.2.2 Buckling Analysis](on page 2353)
- [A. To specify buckling analysis](on page 932)
- [G.17.2.2 Buckling Analysis](on page 2353)
**TR.37.5 Direct Analysis**

**General Format**

```plaintext
PERFORM DIRECT ANALYSIS ( (LRFD | ASD) TAUTOL f1 DISPTOL f2 ITERDIRECT i3 (REDUCEDEI i4) (PDITER 15) (TBITER) PRINT print-options )
```

`print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL }`

This command directs the program to perform the analysis that includes:

a. Reduce Axial & Flexure stiffness to 80% for members selected in the Define Direct input. The 80% applies only to analysis.

b. Solving the static case which has notional loads included.

c. Perform iterations of the iterative PDelta with SmallDelta analysis procedure (default 15 iterations).

d. Solving simultaneous equations for displacements;

e. Compute Tau-b of AISC 05 Direct Analysis Appendix 7 based on required strength versus yield strength.

f. Reforming the global joint stiffness matrix.

g. Solving simultaneous equations for displacements;

h. Repeat steps c) through g) until converged or ITERDIRECT iterations are reached.

A Direct Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification ([TR.32.11 Repeat Load Specification](on page 2770)) and/or REFERENCE LOAD specification ([TR.33 Reference Load Cases - Application](on page 2789)). Direct analysis will not be performed for LOAD COMBINATIONS cases.

Notional loads must be defined using a DEFINE NOTIONAL table.

A list of members which will have their initial Tau-b value set and/or have their Axial stiffness reduced and/or their flexural stiffness reduced must be entered using a DEFINE DIRECT table.

For information on NOTIONAL LOADS, see sections [TR.31.7 Definition of Direct Analysis Members](on page 2643) and [TR.32.14 Notional Loads](on page 2785).

PDELTA iterative load adjustments are computed for frame members only. They are not calculated for plate or solid elements. The results are based on “P-large&small Delta” effects (refer to Option 1 in [TR.37.2 P-Delta Analysis Options](on page 2797)).

Convergence occurs when 2 consecutive iterations have all member tau-b values the same within a tolerance, TAUTOL, and displacements & rotations the same within a tolerance, DISPTOL.

LRFD is the default (all generated loads are factored by 1.0). If ASD entered then loads are factored by 1.6 for the Pdelta and Tau-b calculations. ASD final results are based on the final displacements divided by 1.6.

If resulting displacements are diverging, then the P-Delta iterations will be terminated and the current iteration results will be used as the final results for that load case.

**Table 273: Direct analysis parameters**

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TAUTOL $f_1$</td>
<td>0.01</td>
<td>Tau-b tolerance $f_1$ is normally 0.001 to 1.0.</td>
</tr>
</tbody>
</table>
### Parameter Description

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISPTOL (f_2)</td>
<td>0.01 inch (displacement) 0.01 radians (rotation)</td>
<td>Displacement tolerance (f_2) should not be too tight. The value is in current length units.</td>
</tr>
<tr>
<td>ITERDIRECT (i_3)</td>
<td>1</td>
<td>Limits the number of iterations. A value for (i_3) between 1 to 10 is typically sufficient.</td>
</tr>
<tr>
<td>REDUCEDEI (i_4)</td>
<td>1</td>
<td>Integer, (i_4), specifies whether to use the reduced EI ((\text{Tau-b} \times 0.8 \times \text{EI})) for member section moment and section displacement.</td>
</tr>
<tr>
<td>PDiter (i_5)</td>
<td>15</td>
<td>The number of iterations, (i_5), used in the iterative PDelta with SmallDelta analysis procedure within Direct Analysis; 5 to 25 iterations is the normal range. The default is recommended.</td>
</tr>
<tr>
<td>TBITER</td>
<td>-</td>
<td>If this command is present, then the analysis procedure will iterate Tau-b.</td>
</tr>
</tbody>
</table>

**Note:** You can use the `SET NOPRINT DIRECT` command to turn off the tau-b details in the output file when running a Direct Analysis. This can greatly reduce the volume of output content for models with many load cases.

**Note:** Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.

#### Example

```
PERFORM DIRECT ANALYSIS LRFD TAUTOL 0.01 - DISPTOL 0.01 ITERDIRECT 2 TBITER - PRINT LOAD DATA
```

#### Related Links
- [G.17.2.1.4 AISC 360 Direct Analysis](on page 2352)
- [A. To specify a direct analysis](on page 929)
- [G.17.2.1.4 AISC 360 Direct Analysis](on page 2352)

### TR.37.6 Steady State and Harmonic Analysis

The options available under steady state analysis in STAAD.Pro are described in the next few sections.

#### Related Links
TR.37.6.1 Purpose

This analysis type is used to model steady, harmonically varying load on a structure to solve for the steady harmonic response after the initial transient response has damped out to zero. STAAD.Pro Steady State analysis options include results for one forcing frequency or for a set of frequencies. You may specify ground motion or a distributed joint loading in one load case. Damping is required either in this input or from the Modal Damp input or from the Composite Damping input.

This command is used to specify the analysis request, specify that the load case with the MODAL CALCULATION command (which must be prior to this analysis command) be used as the definition of the mass distribution, and to begin a block of data input describing the steady state forcing functions, the output frequencies, and the printing of the joint responses.

All of the input and output frequencies are in Hertz (Hz or CPS).

Related topics can be found in the following sections:

- G.17.3 Dynamic Analysis (on page 2362)
- TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2538)
- TR.34 Frequency Calculation (on page 2789)
- TR.26.4 Modal Damping Information (on page 2510)
- TR.26.5 Composite Damping for Springs (on page 2511)
- TR.37 Analysis Specification (on page 2795)
- TR.34 Frequency Calculation (on page 2789)

Note: The Modal Calculation command is required in the weight/mass definition load case.

General Format

PERFORM STEADY STATE ANALYSIS

This command directs the program to perform the analysis that includes:

a. Checking whether all information is provided for the analysis;

b. Forming the joint stiffness matrix;

c. Solving simultaneous equations;

d. Solving for modes and frequencies;

e. Computing for the steady state joint displacements, velocities & accelerations and phase angles;

f. Computing the above quantities versus frequency and displaying the results graphically.

g. Member & element forces & stresses and support reactions are not currently computed.

The first input after the Perform Steady State Analysis command is:

BEGIN ( { STEADY | HARMONIC } ) ( { FORCE | GROUND } )

Steady or Harmonic  Steady = the analysis is at one forcing frequency.

Force or Ground  Choose whether the loading is a distributed joint force load or a ground motion.
This command selects which of the 4 load/analysis types, that are available, will be used in this analysis. These four are described in sections TR.37.6.4 Steady Ground Motion Loading (on page 2809) through TR.37.6.7 Harmonic Force Loading (on page 2813).

This block of data should be terminated with the END STEADY command as mentioned in TR.37.6.9 Last Line of this Steady State/Harmonic Analysis (on page 2817).

The steady state/harmonic analysis will calculate the maximum displacement and the associated phase angle for each of 6 joint directions, relative to the ground motion, for each frequency defined in TR.37.6.2 Define Harmonic Output Frequencies (on page 2809).

In PRINT JOINT DISP and in Post processor displayed results, the load case displacement for a given joint and direction will be the maximum value over all of the frequencies (without the phase angles) for a Steady State load case.

In post-processing for harmonic analysis, Log-Log graphs of any joint’s relative translational displacement or velocity or acceleration versus frequency may be selected.

See TR.37.6.8 Print Steady State/Harmonic Results (on page 2815) for printing displacements with phase angles by frequency.

TR.37.6.2 Define Harmonic Output Frequencies

If Harmonic is requested above, then optionally include the next input.

**FREQUENCY (FLO f1 FHI f2 NPTS f3 (MODAL ) FLIST freqs )**

Where:

- **FLO f1** Lowest frequency to be included in Harmonic output. Default to half the first natural frequency.
- **FHI f2** Highest frequency to be included in Harmonic output. Default to highest frequency plus largest difference between two consecutive natural frequencies.
- **NPTS f3** Number of plot frequencies to be included between natural frequencies. Defaults to 5 (7 if fewer than 10 modes) (3 if more than 50 modes). These points are added to improve the graphic display of responses versus frequency.

  - The natural and forcing frequencies are automatically included in the plot frequencies.
- **MODAL** This option causes the natural frequencies between FLO and FHI to be added to the list of forcing frequencies.
- **FLIST freqs** List of forcing frequencies to be included in the Harmonic analysis. Continue freqs input to additional lines by ending each line except the last with a hyphen.

  - Only forcing frequencies will be used to create load case results and print results.

TR.37.6.3 Define Load Case Number

The load case which contains the MODAL CALC command is automatically used for the load case number.

TR.37.6.4 Steady Ground Motion Loading

This set of commands may be used to specify steady ground motion loading on the structure, the ground motion frequency, the modal damping, and the phase relationship of ground motions in each of the global directions.
General Format

This command specifies the ground motion frequency and damping.

\texttt{STEADY GROUND FREQ f1 \{ DAMP f2 | CDAMP | MDAMP \} \{ ABSOLUTE | RELATIVE \}}

Where:

- \texttt{FREQ f1} the steady state frequency in cycles/sec at which the joint loads below will oscillate.
- \texttt{DAMP f2} Damping ratio for all modes when DAMP is selected. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP, MDAMP, and CDAMP select source of damping input:

- \texttt{DAMP} indicates to use the \texttt{f2} value for all modes
- \texttt{MDAMP} indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- \texttt{CDAMP} indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

ABSOLUTE or RELATIVE. Ground motion results in output file will be relative to the ground unless ABSOLUTE is specified. Graphical results are relative. This option has no effect for the force loading cases.

General Format

Enter the direction of the ground motion, the acceleration magnitude, and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

\texttt{GROUND MOTION \{ X | Y | Z \} \{ ACCEL | DISP \} f3 PHASE f4}

Where:

- \texttt{f3} = Ground acceleration in g's or displacement in length units.
- \texttt{f4} = Phase angle in degrees

Related Links

- \texttt{G.17.3.3 Damping Modeling} (on page 2366)
- \texttt{G.17.3.3 Damping Modeling} (on page 2366)

TR.37.6.5 Steady Force Loading

This set of commands may be used to specify JOINT loads on the structure, the forcing frequency, the modal damping, and the phase relationship of loads in each of the global directions.

General Format

This command specifies the forcing frequency and damping for a case of steady forces.

\texttt{STEADY FORCE FREQ f1 \{ DAMP f2 | CDAMP | MDAMP \}}

Where:

- \texttt{FREQ f1} the steady state frequency in cycles/sec at which the joint loads below will oscillate.
- \texttt{DAMP f2} Damping ratio for all modes when DAMP is selected. Default value is 0.05 (5% damping if 0 or blank entered).
DAMP, MDAMP, and CDAMP select source of damping input:

- **DAMP** indicates to use the \( f_2 \) value for all modes
- **MDAMP** indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- **CDAMP** indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

**Joint Loads**

Refer to **TR.32.1 Joint Load Specification** (on page 2651) for additional information on Joint Loads

\[
\text{JOINT LOAD ( [ PHASE *{ X | Y | Z } f_7 ] )}
\]

Where:

- **PHASE \( f_7 \)** Phases angle in degrees. One phase angle per global direction.

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of 0.0. All forces specified below will be applied with the phase angle specified above, if any. Default is 0.0.

Next are the joint forces, if any. Repeat as many lines of joint force data as needed.

\[
\text{joint-list *{ FX } f_1 \ | \ FY } f_2 \ | \ FZ \ f_3 \ | \ MX \ f_4 \ | \ MY \ f_5 \ | \ MZ \ f_6 \}
\]

Where:

- **FX \( f_1 \), FY \( f_3 \), FZ \( f_3 \)** specify a force in the corresponding global direction.
- **MX \( f_4 \), MY \( f_4 \), MZ \( f_6 \)** specify a moment in the corresponding global direction.

**Notes**

- a. Joint numbers may be repeated where loads are meant to be additive in the joint.
- b. UNIT command may be on lines in between joint-list lines.
- c. Forces applied at a slave degree of freedom will be ignored.

**Copy Loads**

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

\[
\text{COPY LOAD}
\]

\[
i_1, \ f_1, \ i_2, \ f_2 \ ... \ i_n, \ f_n
\]

Where:

- \( i_1, i_2 \ ... \ i_n \) prior primary load case numbers that are in this analysis set.
- \( f_1, f_2 \ ... \ f_n \) corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

**Related Links**

- **G.17.3.3 Damping Modeling** (on page 2366)
TR.37.6.6 Harmonic Ground Motion Loading

This set of commands may be used to specify harmonic ground motion loading on the structure, the modal damping, and the phase relationship of ground motions in each of the global directions.

Response at all of the frequencies defined in TR.37.6.2 Define Harmonic Output Frequencies (on page 2809) will be calculated.

General Format

\[
\text{HARMONIC GROUND \{ DAMP } f_2 | \text{CDAMP} | \text{MDAMP} \} \{ \text{ABSOLUTE} | \text{RELATIVE} \}
\]

Where:

- **DAMP** \( f_2 \) Damping ratio for all modes when DAMP is selected. Default value is 0.05 (5% damping if 0 or blank entered).

This command specifies the damping. The steady state response will be calculated for each specified output frequency entered or generated, see TR.37.6.2 Define Harmonic Output Frequencies (on page 2809).

DAMP, MDAMP, and CDAMP select source of damping input:

- DAMP indicates to use the \( f_2 \) value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

ABSOLUTE or RELATIVE. Ground motion results in output file will be relative to the ground unless ABSOLUTE is specified. Graphical results are relative. This option has no effect for the force loading cases.

General Format

\[
\text{GROUND MOTION \{ X | Y | Z \} \{ ACCEL | DISP \} f_3 PHASE f_4}
\]

Enter the direction of the ground motion, the acceleration and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

Where:

- \( f_3 \) Ground acceleration in g's or displacement in length units.
- **PHASE** \( f_4 \) Phase angle in degrees

Next is an optional amplitude versus frequency specification to be used when the ground motion acceleration is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the acceleration \( f_3 \) entered above. If no amplitude data is entered for a direction then the acceleration is \( f_3 \) for that direction.

\[
\text{AMPLITUDE ( A a b c )}
\]

where

\[
\begin{align*}
\text{Amplitude} & = a \times \omega^2 + b \times \omega + c \\
\omega & = \text{forcing frequency in rad/sec.} \\
a, b, c & = \text{Constants. a and b default to 0.0 and c defaults to 1.0.}
\end{align*}
\]
Or

AMPLITUDE

(f1 a1 f2 a2 ... fn an )

(f1 a1 f2 a2 ... fn an ) Frequency-Amplitude pairs are entered to describe the variation of acceleration with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.

One Ground Motion and Amplitude command set can be entered for each global direction.

Related Links
- G.17.3.3 Damping Modeling (on page 2366)
- G.17.3.3 Damping Modeling (on page 2366)

TR.37.6.7 Harmonic Force Loading

This set of commands may be used to specify JOINT loads on the structure, the modal damping, and the phase relationship of loads in each of the global directions.

Response at all of the frequencies defined in TR.37.6.2 Define Harmonic Output Frequencies (on page 2809) will be calculated.

General Format

HARMONIC FORCE { DAMP f2 | CDAMP | MDAMP }

Where:

DAMP f2 Damping ratio for all modes when DAMP is selected. Default value is 0.05 (5% damping if 0 or blank entered).

This command specifies the damping for a case of harmonic forces.

DAMP, MDAMP, and CDAMP select source of damping input:

- DAMP indicates to use the f2 value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

Joint Loads

Refer to TR.32.1 Joint Load Specification (on page 2651) for additional information on Joint Loads

JOINT LOAD ( [ PHASE *{ X | Y | Z } f7 ] )

Where:

PHASE f7 Phase angle in degrees. One phase angle per global direction.

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of 0.0. All forces specified below will be applied with the phase angle specified above, if any. Default is 0.0.
Next are the joint forces, if any. Repeat as many lines of joint force data as needed.

\[
\text{joint-list } \{*\ FX \ f1 \ | \ FY \ f2 \ | \ FZ \ f3 \ | \ MX \ f4 \ | \ MY \ f5 \ | \ MZ \ f6 \ *
\]

Where:

- \(FX f1, FY f3, FZ f3\) specify a force in the corresponding global direction.
- \(MX f4, MY f4, MZ f6\) specify a moment in the corresponding global direction.

\section*{Notes}

\begin{enumerate}
\item Joint numbers may be repeated where loads are meant to be additive in the joint.
\item UNIT command may be on lines in between joint-list lines.
\item Forces applied at a slave degree of freedom will be ignored.
\end{enumerate}

\section*{Copy Loads}

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

\begin{verbatim}
COPY LOAD
i1, f1, i2, f2 ... in, fn
\end{verbatim}

Where:

- \(i1, i2 ... in\) prior primary load case numbers that are in this analysis set.
- \(f1, f2 ... fn\) corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

Next is an optional amplitude versus frequency specification to be used when the ground motion acceleration is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the acceleration \(f3\) entered above. If no amplitude data is entered for a direction then the acceleration is \(f3\) for that direction.

\begin{verbatim}
AMPLITUDE ( A a B b C c )
\end{verbatim}

where

- \(\text{Amplitude} = a\omega^2 + b\omega + c\)
- \(\omega = \) forcing frequency in rad/sec.
- \(a, b, c = \) Constants. \(a\) and \(b\) default to 0.0 and \(c\) defaults to 1.0.

Or

\begin{verbatim}
AMPLITUDE
(f1 a1 f2 a2 ... fn an )
\end{verbatim}

- \(f1 a1 f2 a2 ... fn an\) Frequency - Amplitude pairs are entered to describe the variation of acceleration with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.
Leaving the direction field blank or inserting ALL will use the same Frequency versus Amplitude for all 3 force directions.

Enter amplitudes for up to three directions. For directions without amplitude input, including moment directions, the amplitude will be set to 1.0.

The first form of the amplitude input is converted to the second form by creating 199 equally spaced frequencies, then computing the amplitude for each frequency.

**TR.37.6.8 Print Steady State/Harmonic Results**

**General Format**

```plaintext
PRINT HARMONIC DISPLACEMENTS list-spec
```

where

`list-spec = { (ALL) | LIST list of items-joints }`

This command must be after all steady state/harmonic loadings and before the `END STEADY` command. For each harmonic frequency of TR.37.6.2 Define Harmonic Output Frequencies (on page 2809) the following will be printed:

1. Modal responses.
2. Phase angles with 1 line per selected joint containing the phase angle for each of the 6 directions of motion.
3. Displacements table with 1 line per selected joint containing the maximum displacements for each of the 6 directions of motion.
4. Velocities.
5. Accelerations.

**Steady State Notes**

For members, the final results at the joints will be complete but the section results will be as if the member loads were applied as statically equivalent loads at the member ends.

**Steady State Examples**

**Example 1**

```plaintext
BEGIN STEADY GROUND
STEADY GROUND FREQ 22.4 DAMP .033 ABS
GROUND MOTION X ACC .11 PHASE 0.0
GROUND MOTION Y ACC .21 PHASE 10.0
GROUND MOTION Z ACC .15 PHASE 20.0
PRINT HARMONIC DISP ALL
END
```

**Example 2**

```plaintext
BEGIN HARMONIC GROUND
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 - 17 21 30
HARMONIC GROUND DAMP .033 REL
GROUND MOTION X ACC .11 PHASE 0.0
AMPLIT A 0.10 B .21 C .03
GROUND MOTION Y DIS .21 PHASE 10.0
AMPLITUDE
3 5 5 4 10 6 -
35 3
```
**Example 3**

BEGIN STEADY FORCE  
STEADY FORCE FREQ 11.2 DAMP .033  
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0  
UNIT KIP  
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0  
UNIT POUND  
10 5 TO 7 BY 2 - 88 FX 10.0 FY 5.0  
COPY LOAD  
1 1.5 2 0.8 - 3 1.0  
PRINT HARMONIC DISP ALL  
END

**Example 4**

BEGIN HARMONIC FORCE  
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 - 17 21 30  
HARMONIC FORCE DAMP .033  
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0  
UNIT KIP  
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0  
UNIT POUND  
10 5 TO 7 BY 2 - 88 FX 10.0 FY 5.0  
COPY LOAD  
1 1.5 2 0.8 - 3 1.0  
AMPLIT X A 0.10 B .21  
AMPLITUDE Y 3 5 5 4 10 6 - 35 3  
AMPLIT Z A 0.10 C 0.03  
PRINT HARMONIC DISP ALL  
END

**Example 5**

BEGIN HARMONIC FORCE  
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 - 17 21 30  
HARMONIC FORCE DAMP .033  
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0  
UNIT KIP  
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0  
UNIT POUND  
10 5 TO 7 BY 2 - 88 FX 10.0 FY 5.0  
COPY LOAD  
1 1.5 2 0.8 - 3 1.0  
AMPLIT ALL A 0.10 B .21
TR.37.6.9 Last Line of this Steady State/Harmonic Analysis

Used to end a Steady State or Harmonic Analysis.

### General Format

```
END STEADY
```

### TR.37.6.10 Steady State Examples

**Note:** For full examples, refer to the files SVM33.std, SVM32.std, SS-beam2.std, SS-beam3.std, Exam07.std, and Exam14b.std in the folder C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\SteadyState\.

**Example 1: Steady Ground Motion**

```
BEGIN STEADY GROUND
STEADY GROUND FREQ 22.4 DAMP .033 ABS
GROUND MOTION X ACC .11 PHASE 0.0
GROUND MOTION Y ACC .21 PHASE 10.0
GROUND MOTION Z ACC .15 PHASE 20.0
PRINT HARMONIC DISP ALL
END
```

**Example 2: Harmonic Ground Motion**

```
BEGIN HARMONIC GROUND
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 - 17 21 30
HARMONIC GROUND DAMP .033 REL
GROUND MOTION X ACC .11 PHASE 0.0
AMPLIT A 0.10 B .21 C 0.03
GROUND MOTION Y DIS .21 PHASE 10.0
AMPLITUDE 3 5 5 4 10 6 - 35 3
GROUND MOTION Z ACC .15 PHASE 20.0
AMPLIT A 0.10 B .21 C 0.03
PRINT HARMONIC DISP ALL
END
```
Example 3: Steady Forcing Frequency

BEGIN STEADY FORCE
STEADY FORCE FREQ 11.2 DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
PRINT HARMONIC DISP ALL
END

Example 4: Harmonic Forcing Frequency with Directional Amplitudes

BEGIN HARMONIC FORCE
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC FORCE DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT X A 0.10 B .21
AMPLITUDE Y
3 5 5 4 10 6 -
35 3
AMPLIT Z A 0.10 C 0.03
PRINT HARMONIC DISP ALL
END

Example 5: Harmonic Forcing Frequency with Same Amplitudes

BEGIN HARMONIC FORCE
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC FORCE DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT ALL A 0.10 B .21
TR.37.7 Pushover Analysis

This section is a Technical Reference on the input parameters used to define a pushover analysis in STAAD.Pro.

Related Links
- Perform Pushover Analysis tab (on page 3075)
- A. To specify a pushover analysis (on page 933)

TR.37.7.1 Beginning of Pushover Data

All Pushover analysis related data are to be entered after entering this command.

General Format

```
DEFINE PUSHOVER DATA
```

TR.37.7.2 Define Input

The following sections describe the input parameters used for a pushover definition.

TR.37.7.2.1 Type of Frame

FEMA recognizes only three types of frames: concentric braced frame (CBF), eccentric braced frame (EBF), and moment frame. Only CBF and fully restrained (FR) moment frame are used in the program.

General Format

```
FRAME i1
```

where:

```
i1    Type of frame:
  1. for a Concentric Braced frame
  2. for a Moment frame (default)
```

Related Links
- G.17.4.2.1.1 Define Steel Moment and Braced Frames (on page 2386)
- M. To define general pushover data (on page 862)

TR.37.7.2.2 Expected Yield Stress

Expected yield stress may be specified.

General Format

```
FYE f1 MEMBER { <list> | ALL }
```

Technical Reference of STAAD Commands

TR.37 Analysis Specification

where:

\[ f_1 \] User-defined expected yield stress (default = 50 ksi). Refer to Section 5.3.2.3 and Tables 5-1 and 5-2 of FEMA 356 for additional information on values used.

\[ <\text{list}> \] List of members associated with the user defined yield stress

Related Links
- M. To define member specific pushover data (on page 862)

TR.37.7.2.3 Effective Length Factor of Member

The effective length factor of any member can be defined. This value is used in finding Euler Buckling load of a member. The minimum of these values two will be taken in calculation.

**General Format**

\[
\begin{align*}
\text{KY} & \quad f_2 & \quad \text{MEMBER} \{ \quad <\text{list}> \quad | \quad \text{ALL} \quad \} \\
\text{KZ} & \quad f_3 & \quad \text{MEMBER} \{ \quad <\text{list}> \quad | \quad \text{ALL} \quad \}
\end{align*}
\]

where:

\[ f_2 \] User-defined K value in local y-axis (default = 1.0)

\[ f_3 \] User-defined K value in local z-axis (default = 1.0)

\[ <\text{list}> \] List of members associated with the user-defined effective length factor

Related Links
- M. To define member specific pushover data (on page 862)

TR.37.7.2.4 Consideration of Geometric Nonlinearity Effect

The following command is used to optionally consider geometric nonlinear effects. You may specify either (or both) the DISPTOL or GEOCYCLE commands to optionally check for convergence.

**General Format**

\[
\begin{align*}
\text{GNONL} & \quad i_2 \\
( \quad \text{DISPTOL} & \quad f_4 ) \\
( \quad \text{GEOCYCLE} & \quad i_3 )
\end{align*}
\]

\[ i_2 \] Specifies if geometric nonlinear effects are to be considered

- 0. = Include the effect of large displacements (Default)
- 1. = Ignore the effect of large displacements

\[ f_4 \] Convergence displacement tolerance for convergence of Geometric Nonlinearity, in current units of length. The default value is the maximum span of the structure divided by 120.

See Note a below.
i3  Number of iterations to be performed for convergence of Geometric Nonlinearity. Default value is one.

See Note b below.

Notes

a. The member end forces are evaluated by performing a convergence check on the joint displacements. In each step, the displacements are compared with those of the previous iteration in order to check whether convergence is attained.

Refer SET DISPLACEMENT command in TR.5 Set Command Specification (on page 2413) for additional details. If any displacement on any iteration exceeds the DISPTOL limit, the solution is diverging and is terminated.

b. If the number of nonlinear iterations exceeds GEOCYCLE limit the analysis is terminated. Please refer NONLINEAR ANALYSIS in STAAD for more details. However, maximum allowable iteration is 50. If the number of iterations exceeds 50 the analysis is terminated.

Related Links

• M. To define general pushover data (on page 862)

TR.37.7.2.5 KG Matrix Iteration

Optional control to limit the number of iterations performed on the geometric stiffness, KG, matrix.

General Format

IKGITER  i4

where:

i4  The number of iterations to be performed for KG matrix iterations. If no value is specified, the program will assume the value of GEOCYCLE (on page 2820).

If the value is one (1), the program will continue iteration until the solution converges. However, maximum allowable iteration is 50. If the number of iterations exceeds 50, the analysis is terminated.

Related Links

• M. To define general pushover data (on page 862)

TR.37.7.2.6 Maximum number of Analysis cycle

Used to specify a maximum number of cycles to be considered during the strain hardening stage for the load increment stage.

Whenever there is load increment on the structure, a new analysis cycle starts. This consists of one analysis cycle for gravity loading and the rest consists of the sum total of number of cycles required in the linear stage plus the number of cycles required to pass the structure from the linear to the nonlinear and strain hardening stage plus the number of cycles required in the strength degraded stage. The strength degraded stage consists of both analysis cycles during the load increment and load decrement stages. You can specify a lower number of analysis cycles to be considered during the strain hardening stage for the load increment stage only, which will be considered after pushover analysis enters into the nonlinear stage.
**Note:** The sum total of number of cycles specified by user plus number of analysis cycles required in load decrement stage plus number of cycles required in linear stage including analysis cycle for gravity loading will be limited to 10001 cycles.

**General Format**

```
MAXCYCLE i5
```

where:

- `i5` Number of analysis cycles.

**Related Links**

- [M. To define general pushover data](on page 862)

**TR.37.7.2.7 Print Output Result**

Only the final state is saved for nonlinear static analysis. You may instruct the program to print analysis result (joint displacements, member end forces, and support reactions) for the final state of nonlinear static analysis in the output file.

**General Format**

```
PRINT RESULT ( OUTPUT f5 )
```

where:

- `f5` Optionally used to limit the output to displacements, member end forces, or reactions.
  1. Joint displacements only
  2. Member end forces only
  3. Support reactions only

**Related Links**

- [M. To define general pushover data](on page 862)

**TR.37.7.2.8 Save Output Results for Multiple Steps**

By default the intermediate analysis results are saved for positive increments only at 0.1-inch incremental value of displacement at roof or at control joint. However, you instruct the program to save results at specific intermediate steps. This will create binary post-processing files that include load step results at user-defined points.

**General Format**

```
SAVE LOADSTEP RESULT { DISP | BSHEAR } f6
```

- `f6` If DISP is defined, this is the incremental value of displacement at roof or at control joint in current units. If no value is given 0.1 inch is taken as default.
  - If BSHEAR is defined, this is the incremental value of base shear in current units. If no value is given, 5.0 kip is taken as default.
If the value of base shear defined is $B$ and incremental value of $\text{BSHEAR}$ is $f$, the number of intermediate results that can be saved in $B / f$. The maximum allowable value is 500. Similarly, if the value of allowable displacement at control joint is $D$ and incremental value of $\text{DISP}$ is $n$, the number of intermediate results that can be saved in $D / n$. The maximum allowable value is 500.

Besides saving user-defined intermediate results, the other cases where intermediate results will be saved are as follows:

- A frame hinge is formed
- A frame hinge is trying to unload
- A frame hinge has failed

**Related Links**

- *M. To define general pushover data* (on page 862)

**TR.37.7.3 Define Loading Pattern**

The pattern of the push load distribution on the nodes of the structure to be entered.

**General Format**

```
LOADING_PATTERN i1
```

where:

- $i1$ Method of loading pattern used:
  - 0. = STAAD calculates internally the push load based on the specific gravity load and the first modal displacement in the direction of push load (Default)
  - 1. = user-defined push load pattern

**Note:** You can define the Push load pattern explicitly when the \text{LOADING\_PATTERN} command value is one. The external push load is defined as primary load case. See *TR.37.7.8 Pushover Loading Input* (on page 2829).

**Related Links**

- *G.17.4.2.1.3 Define Lateral (Push) Loading* (on page 2387)
- *G.17.4.1.8 Lateral Load Distribution* (on page 2385)
- *M. To add a pushover loading* (on page 864)

**TR.37.7.3.1 Program Defined Push Load Distribution Pattern**

If \text{LOADING\_PATTERN} is 0 (i.e., STAAD.Pro internally calculates the lateral push load), you are required to define following two inputs.

For all analyses, at least two vertical distributions of lateral load shall be applied. One pattern shall be selected from each of the following two groups. Therefore, two different input files are to be generated. One analysis is to be performed by selecting any one method from Group 1. Another separate analysis is to be performed by selecting Method 3 from Group 2.
**Group 1**

**Method 1** A vertical distribution proportional to the value of $C_{\text{ex}}$ given in Equation (3-12) of FEMA 356:2000 is performed. Use of this distribution shall be permitted only when more than 75% of the total mass participates in the fundamental mode in the direction under consideration, and the uniform distribution is also used.

**Method 2** A vertical distribution proportional to the shape of the fundamental mode in the direction under consideration is performed. Use of this distribution shall be permitted only when more than 75% of the total mass participates in the fundamental mode in the direction under consideration, and the uniform distribution is also used.

**Group 2**

**Method 3** A vertical distribution is performed consisting of lateral forces at each level proportional to the total mass at each level.

**General Format**

- **VDB i2**

  - **i2** Method used for load distribution pattern:
    1. Method 1
    2. Method 2
    3. Method 3

**Related Links**

- **M. To add a pushover loading** (on page 864)

**TR.37.7.3.2 Total Base Shear to be Distributed**

Optionally used to define base shear that will be distributed vertically along the height of the structure at each floor level.

**Note:** If base shear to be distributed is not defined, the program distributes 10 percent of gravity loading as lateral load.

**General Format**

- **DISTRIBUTE BASE SHEAR { X | Z } f1**

  - **f1** Total base shear to be distributed in current units of force.

**Related Links**

- **M. To add a pushover loading** (on page 864)

**TR.37.7.3.3 Number of Push Load Steps**

This command is used to specify number of steps for the push load.
The lateral load at each floor divided by the number of load steps gives the push load increment at that floor level.

**General Format**

```
LDSTEP i3
```

- **i3** Number of load steps. Default is 100.

**Related Links**
- [M. To add a pushover loading](#) (on page 864)

**TR.37.7.4 Define Solution Control**

One of the two following methods must be specified to define the limit of the pushover analysis.

**Note:** Both method may be specified in the same input file. Upon exceeding either of the limits, the analysis will stop.

**Related Links**
- [G.17.4.2.1.6 Define Pushover Analysis Solution Control](#) (on page 2390)
- [M. To define solution control](#) (on page 865)

**TR.37.7.4.1 Push Up to Defined Base Shear**

The pushover analysis will continue until the cumulative base shear is less than or equal to the base shear specified by this command or the structure has additional strength.

**General Format**

```
BASE SHEAR { X | Z } DEFINED f1
```

- **f1** Explicitly defined base shear in current units of force.

**TR.37.7.4.2 Push Up to Defined Displacement at Control Joint**

The pushover analysis will continue until the displacement at the specified joint at the specified direction exceeds specified displacement.

**General Format**

```
DISP { X | Z } f2 JOINT i1
```

- **f2** Explicitly defined joint displacement value in current units of length.
- **i1** Node number of control joint.

**TR.37.7.5 Define Hinge Property**

If the automatic hinge calculation per FEMA 356 is not used, then the hinge properties must be defined and these hinges must be assigned to members.
**TR.37.7.5.1 User-Defined Hinge Property**

Several hinge property may be specified with specific type identifier. User may later refer this type identifier while assigning specific hinge properties to members (See “TR.37.7.9 Validation of Commands other than Input Parameters” (on page 2829)). If hinge property is not specified, STAAD assumes the FEMA specified hinge properties.

To specify uncoupled moment hinge property of the member following input is required.

**General Format**

<table>
<thead>
<tr>
<th>HINGE PROPERTY</th>
<th>MOMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>TYPE n1</td>
<td></td>
</tr>
<tr>
<td>A x1 y1 B yb1 y1 C xc1 yc1 D xd1 yd1 E xe1 ye1 YM f11 YR f21</td>
<td></td>
</tr>
<tr>
<td>IO f31 LS f41 CP f51</td>
<td></td>
</tr>
<tr>
<td>...</td>
<td></td>
</tr>
<tr>
<td>TYPE ni</td>
<td></td>
</tr>
<tr>
<td>A xai yai B xbi ybi C xci yci D xdi ydi E xei yei YM f1i YR f2i</td>
<td></td>
</tr>
<tr>
<td>IO f3i LS f4i CP f5i</td>
<td></td>
</tr>
<tr>
<td>n1,..., ni</td>
<td>Type identifier (Hinge property number)</td>
</tr>
<tr>
<td>A, B, C, D, E</td>
<td>Points on the load deformation curve at points A, B, C, D, and E as specified in FEMA</td>
</tr>
<tr>
<td>xa1 y1,..., xe1 ye1,..., xai yai,..., xei yei</td>
<td>Coordinates of A, B, C, D, E respectively</td>
</tr>
<tr>
<td>xa1,..., xe1,..., xai,..., xei</td>
<td>Deformation Ratio, ( \theta/\theta_y )</td>
</tr>
<tr>
<td>ya1,..., ye1,..., yai,..., yei</td>
<td>Moment Ratio, ( Q/Q_y )</td>
</tr>
<tr>
<td>f11,...,f1i</td>
<td>Yield moment in current force units</td>
</tr>
<tr>
<td>f21,...,f2i</td>
<td>Yield rotation in radians</td>
</tr>
<tr>
<td>f31,...,f3i</td>
<td>Immediate occupancy in deformation ratio</td>
</tr>
<tr>
<td>f41,...,f4i</td>
<td>Life safety in deformation ratio</td>
</tr>
<tr>
<td>f51,...,f5i</td>
<td>Collapse prevention in deformation ratio</td>
</tr>
</tbody>
</table>

The coordinates of point A should be always (0, 0) and that of point B is (1.0, 1.0). Both the coordinates of point C must be greater than 1.0. The X coordinate of point D must be same as that of point C and Y coordinate must be less than 1.0. The X coordinate of point E must be greater than that of point D and Y coordinate should be same as that of point D.

**Note:** The hinge property will be applied to hinge formed due to moment about local z-axis for beam. For column, hinge formation will be considered either about local y-axis or about local z-axis depending upon whichever moment is dominating and also the orientation of the column with respect to the push load direction. The hinge property will be applied to the hinge formed due to this moment.

See “TR.37.7.5.2 Assignment of Hinge Property to the Members” (on page 2827)” for example input.

**Related Links**

- Technical Reference of STAAD Commands
- TR.37 Analysis Specification
- STAAD.Pro User Manual
TR.37.7.5.2 Assignment of Hinge Property to the Members

This command is used to define the method by which the plastic hinges are formed. If no assignment of hinge property is done, the program will assign built-in FEMA 356 hinge property to the members.

**HINGE**

```
{ TYPE n1 | FEMA | IGNORE } { MEMBER <list> | ALL }
```

- **n1**: Hinge type identifier
- **<list>**: List of members associated the current hinge command

**Notes**

a. Once **HINGE PROPERTIES** are specified (See "**TR.37.7.5.1 User-Defined Hinge Property**" (on page 2826)), you can assign these to the members using the **HINGE TYPE** command.

**Note**: Hinge properties must be defined prior to assigning them to members in the input file.

b. If a user-defined hinge property (i.e., the **HINGE TYPE** command) is assigned to some members, the program will consider hinge formation in these members only. The **HINGE FEMA** hinge property must be assigned to other members. Thus, you must take care to assign hinge property to all members if hinge formation is to be considered in all members.

In this case, the **HINGE FEMA** command must follow after the **HINGE TYPE** command in the input.

c. The **HINGE IGNORE** command can be used to ignore hinge formation in some members.

**Note**: The ALL option should not be used with **HINGE IGNORE** as this will prevent a successful pushover analysis.

**Example**

```
DEFINE PUSHOVER DATA
HINGE PROPERTY MOMENT
TYPE 1
A 0.0 0.0 B 1.0 1.0 C 10.0 1.2 D 10.0 0.6 E 12.0 0.6 YM 3325 YR 0.00899
LO 1.001 LS 6.0 CP 8.0
...
HINGE IGNORE MEMBER 16
HINGE TYPE 1 MEMBER 1 3 17
HINGE FEMA MEMBER 2 4 to 15 18 to 20
...
END PUSHOVER DATA
```

**Related Links**

- **G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria** (on page 2388)
- **G.17.4.1.4 Types of Nonlinearity** (on page 2381)
- **G.17.4.1.6 Frame element hinge properties** (on page 2382)
- **M. To manually define and assign hinges** (on page 863)
TR.37.7.6 Define Spectral Parameters

The parameters of this section are used to construct the response spectrum according to FEMA 356:2000.

**General Format**

```
SPECTRUM PARAMETERS
DAMPING f1 ( f2 f3 f4 )
SC f5
SS f6
S1 f7
```

- **f1** Percentage of critical damping for the 1st response spectrum (Default 5%)
- **f2** Optional percentage of critical damping for the 2nd response spectrum (Default is zero)
- **f3** Optional percentage of critical damping for the 3rd response spectrum (Default is zero)
- **f4** Optional percentage of critical damping for the 4th response spectrum (Default is zero)
- **f5** An integer corresponding to the site class as described in the following table. The default value is 4 (i.e., site class D):

<table>
<thead>
<tr>
<th>Site Class</th>
<th>Value of f5</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1</td>
</tr>
<tr>
<td>B</td>
<td>2</td>
</tr>
<tr>
<td>C</td>
<td>3</td>
</tr>
<tr>
<td>D</td>
<td>4</td>
</tr>
<tr>
<td>E</td>
<td>5</td>
</tr>
</tbody>
</table>

- **f6** Spectral acceleration at short-period, $S_s$
- **f7** Spectral acceleration at one-second period, $S_1$

**Related Links**
- **G.17.4.2.1.7 Define Input for Demand Spectrum** (on page 2390)
- **M. To define pushover spectral data** (on page 863)

TR.37.7.7 End Pushover Data

All Pushover data are to be entered before this command.

**General Format**

```
END PUSHOVER DATA
```
Tip: This command is added automatically when the DEFINE PUSHOVER DATA command is added through the Create New Definitions / Load Cases / Load Items dialog in the graphical environment.

Related Links
- M. To define general pushover data (on page 862)

TR.37.7.8 Pushover Loading Input

Only two types of loading are accepted in pushover analysis.

Refer to TR.32 Loading Specifications (on page 2650) for additional information on primary load types.

Gravity Load

Syntax for gravity loading is as follows:

```
LOAD i1 LOADTYPE GRAVITY ( TITLE any_load_title )
```

i1 gravity loading number

Push Load

Syntax for push loading (user-defined external incremental push loading pattern on the structure) is as follows:

```
LOAD i2 LOADTYPE PUSH ( TITLE any_load_title )
```

i2 user-defined push loading number

Note: For user-defined incremental push load, the loading has to be applied only in global X or Z directions. Combinations of X and Z directions is not accepted. Also, the lateral load in either global X or Z directions is to be applied in form of joint loads.

Related Links
- G.17.4.2.1.3 Define Lateral (Push) Loading (on page 2387)
- G.17.4.1.8 Lateral Load Distribution (on page 2385)
- M. To add a pushover loading (on page 864)

TR.37.7.9 Validation of Commands other than Input Parameters

1. The LOADTYPE is required and must be either GRAVITY or PUSH. See “TR.37.7.8 Pushover Loading Input” (on page 2829)
2. There can be more than one gravity load case. STAAD internally combines all gravity load cases into one. Analysis uses this combined gravity loading only.
3. A mass vector for eigen solution is formed from this combined gravity loading. There is no need to provide mass modeling.
4. Only one push load case (either in the form of user defined external push load or generated by the program) is allowed. The direction of loading is either global X or global Z. Combination of these directions are not allowed. For user defined incremental push load pattern, only JOINT LOADs in FX or FZ directions may be specified.
Limitations on Pushover Analysis Input

The following inputs are not allowed in Pushover Analysis:

- **PERFORM PUSHOVER ANALYSIS** cannot be repeated more than once.
- **PERFORM PUSHOVER ANALYSIS PRINT** is not accepted.
- **CHANGE** command is not accepted.
- **LOAD COMBINATION** command is not accepted.
- **SET Z UP** command is not accepted.
- **SET RESTART 1** command will not work (i.e., `<filename>.L17` file will not be created).
- **SURFACE** element is not accepted.
- Material concrete is not considered.
- Non-prismatic section is not considered.
- Curved beam is not considered.
- Cable member is not considered.
- Plate and solid elements is not considered.
- Beta angle other than 0 and 90 degree is not accepted.

**TR.37.8 Geometric Nonlinear Analysis**

**Note:** Effective from STAAD.Pro V8i (release 20.07.05).

**General Format**

```
PERFORM NONLINEAR ANALYSIS (ARC f₁) (ITERATION i₁) (TOLERANCE f₂) (STEPS i₂) (REBUILD i₃) (KG (i₄)) (JOINT_TARGET i₅) (DISPL_TARGET f₃) (PRINT print-specs)
```

The first analysis step must be stable, otherwise use ARC control to prevent instability. The procedure does not use follower loads. Loads are evaluated at the joints before the first step; then those loads translate with the joint but do not rotate with the joint. Equilibrium is computed in the displaced position.

**Note:** The nonlinear analysis command requires the Advanced Analysis Engine package.

The following table describes the parameters available for nonlinear analysis:

**Table 274: Geometric Nonlinear Analysis parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ARC</td>
<td>f₁</td>
<td>0.0</td>
<td>Displacement control. Value is the absolute displacement limit for the first analysis step. If max. displacement is greater than this limit, ARC will calculate a new step size for the first step and a new value for STEPS. Value should be in current length units. ARC = 0 indicates no displacement control.</td>
</tr>
<tr>
<td>ITERATION</td>
<td>i₁</td>
<td>100</td>
<td>Max. Number of iterations to achieve equilibrium in the deformed position to the tolerance specified.</td>
</tr>
<tr>
<td>Parameter</td>
<td>Value</td>
<td>Default Value</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------</td>
<td>-------</td>
<td>---------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>TOLERANCE</td>
<td>f_2</td>
<td>0.0001 inch</td>
<td>For convergence, two successive iteration results must have all displacements the same within this tolerance. Value entered is in current units.</td>
</tr>
<tr>
<td>STEPS</td>
<td>i_2</td>
<td>1</td>
<td>Number of load steps. Load is applied in stages if entered. One means that all of the load is applied in the first step.</td>
</tr>
<tr>
<td>REBUILD</td>
<td>i_3</td>
<td>1</td>
<td>Frequency of rebuilds of the tangent K matrix per load step &amp; iteration.</td>
</tr>
<tr>
<td>KG</td>
<td>i_4</td>
<td>0</td>
<td>This parameter controls whether the geometric stiffness, KG, is added to the stiffness matrix, K.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• KG or KG 1 Use K+KG for the stiffness matrix. (Default)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• KG 0 (or KG omitted) Do not use KG.</td>
</tr>
<tr>
<td>JOINT_TARGET</td>
<td>i_5</td>
<td>none</td>
<td>Joint being monitored in a displacement target analysis.</td>
</tr>
<tr>
<td></td>
<td>i_6</td>
<td>1</td>
<td>Global degree of freedom (1 through 6):</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1. Global X</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2. Global Y</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3. Global Z</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>4. Moment about Global X</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>5. Moment about Global Y</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>6. Moment about Global Z</td>
</tr>
<tr>
<td>DISPL_TARGET</td>
<td>f_3</td>
<td>none</td>
<td>Displacement target value in current length units.</td>
</tr>
<tr>
<td>print-spec</td>
<td></td>
<td>none</td>
<td>Standard STAAD analysis print options.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>See TR.37.1 Linear Elastic Analysis (on page 2796) for details.</td>
</tr>
</tbody>
</table>

If a target displacement is used, then both the JOINT_TARGET and DISPL_TARGET must be defined. If the degree of freedom is not specified, then the Global X direction is assumed.

If Joint target and Displacement target are entered and the STEPS parameter is greater than two; then the analysis will proceed step by step until the targeted joint degree of freedom has displaced DISPL_TARGET length units or more.

Nonlinear entities such as tension/compression members, multilinear springs, gaps, etc. are not supported when using a nonlinear analysis. Additionally, nonlinear analysis does not account for post-buckling stiffness of members.
Caution: The deprecated NONLINEAR n ANALYSIS command will adopt the new procedure (unless a SET NONLINEAR OLD command is entered before the joint coordinates to invoke the old procedure for backward compatibility). This uses n iterations. It is strongly recommended to use the new procedure.

Note: Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.

Example
PERFORM NONLINEAR ANALYSIS ARC 0.01 ITER 200 STEPS 100 TOL 0.00001 - REBUILD 1 KG joint 11 2 DISPL_TARGET 0.5

Related Links
- G.17.2.3 Static Geometrically Nonlinear Analysis (on page 2355)
- A. To specify a nonlinear analysis (on page 930)
- A. To specify a nonlinear analysis (on page 930)
- G.17.2.3 Static Geometrically Nonlinear Analysis (on page 2355)

TR.37.9 Imperfection Analysis

This performs a modified linear elastic analysis using Member Imperfection Specifications (See TR.26.6 Member Imperfection Information (on page 2511)) defined on beam and column members.

An Imperfection analysis will reflect the secondary effects only if the camber and/or drift is specified in a DEFINE IMPERFECTIONS specification (TR.26.6 Member Imperfection Information (on page 2511)). For combination of load cases, with imperfection, use the Repeat Load specification rather than the Load Combination.

General Format
PERFORM IMPERFECTION ANALYSIS (PRINT { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL } )

See TR.37.1 Linear Elastic Analysis (on page 2796) for details.

Without one of these analysis commands, no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

Note: Due to the mechanisms used to include master/slave systems, if the reactions on master nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a master node to the support.

Related Links
- G.17.2.4 Imperfection Analysis (on page 2355)
- A. To specify an imperfection analysis (on page 932)
- G.17.2.4 Imperfection Analysis (on page 2355)
TR.37.10 Floor Spectrum Command

This command is used to specify the calculation of floor spectra from time history acceleration results. The Floor Response Spectrum command must immediately follow an analysis command associated with the time history load case. The acceleration used may be either the absolute acceleration or the relative to ground acceleration.

**Note:** This data should only be entered if there is a time history case being solved.

**General Format**

First line of this Floor Spectrum Data.

```
GENERATE FLOOR SPECTRUM
```

Specify Floor Groups

Each new floor definition starts with the next command:

```
BEGIN FLOOR DIRECTION { GX | GY | GZ }* { TITLE }
```

GX, GY, and GZ specify up to 3 global directions for which acceleration vs frequency spectra will be generated for this floor. Optionally enter a Title/Description (up to 50 characters) for this floor that will be displayed on the graphs in post processing.

The next one or more lines will identify the floors which will have spectrum curves generated either by referencing a NODE GROUP (see TR.16.1 Listing of Entities by Specifying Groups (on page 2440)) or explicitly listing the joints that constitute the floor. For multiple groups, enter each on a separate line.

```
{ _jointgroup | jointlist }
```

Enter as many lines as necessary to specify all of the groups needed to define this floor.

If more floors are to be defined, then repeat the `BEGIN FLOOR DIRECTION` command followed by the `_jointgroup_ name data. Enter as many floors as desired.

After the last floor definition, enter the following parameters that will be used in all of the Floor Spectrum calculations.

```
OPTIONS ( { FLOW f1 | FHIGH f2 | FDELTA f3 | DAMP f4 ( ... fn ) | RELATIVE } ) print-options
```

This command may be continued to the next line by ending the line with a hyphen.

**Where:**

- **FLOW f1** - Lowest frequency to be in the calculated spectrum. The value for FLOW should be at least 0.01 Hz.
- **FHIGH f2** - Highest frequency to be in the calculated spectrum.
- **FDELTA f3** - The spectrum will be calculated at $f_3$ intervals from the FLOW value ($f_1$) to FHIGH ($f_2$).
- **DAMP f4 ... fn** - Up to 10 damping values may be entered. One spectrum will be generated for each damping value for each global direction requested for each floor defined. The spectrum will be based on these modal damping ratios.
- **RELATIVE** - If there is ground motion defined and you want the spectra to be based on the relative acceleration of the floor to the ground acceleration, then enter the relative parameter. Default is absolute.
The **print-options** are:

**THPRINT i1**  Option to print the time history acceleration being used in each spectrum calculation.
- 0 - No print
- 2 - Print the time history accelerations.

**SPRINT**  Include this parameter to print the calculated spectrum.

**Tip:** Omitting these options is recommended for most analysis runs.

Last line of the floor spectrum data:

```
END  FLOOR SPECTRUM
```

---

**Example**

 DEFINE TIME HISTORY
 TYPE 1 FORCE
 0 -20 0.5 100 1 200 1.5 500 2 800 2.5 500 3 70 16 0
 ARRIVAL TIME
 0
 Damping 0.075
 *
 LOAD 1 LOADTYPE SEISMIC TITLE Time History case
 * Mass model required
 SELFWEIGHT X 1
 SELFWEIGHT Y 1
 SELFWEIGHT Z 1
 JOINT LOAD
 1 TO 6 FX 62.223 FY 62.223 FZ 62.223
 * Time loads
 TIME LOAD
 2 FX 1 1
 PERFORM ANALYSIS
 GENERATE FLOOR SPECTRUM
 BEGIN FLOOR DIRECTION GX GZ Ground Motion
 _FL1
 _FL17
 BEGIN FLOOR DIRECTION GX GZ Floor 18 A/C Unit 36
 _FL18
 OPTIONS FLO 0.5 FHI 35.0 FDEL 0.1 –
 DAMP 0.03 0.05 0.07
 END FLOOR SPECTRUM
```

**Note:** A complete example file is installed with the program at

```
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\FloorSpectrum\FloorSpectrumSample.STD (typical location).
```

---

**Related Links**

- **A. To generate a floor spectrum** (on page 935)
- **A. To generate a floor spectrum** (on page 935)
TR.38 Change Specification

This command is used to reset the stiffness matrix. Typically, this command is used when multiple analyses are required in the same run.

General Format

CHANGE

This command indicates that input, which will change the stiffness matrix, will follow. This command should only be used when an analysis has already been performed. The CHANGE command does or requires the following:

• sets the stiffness matrix to zero
• makes members active if they had been made inactive by a previous INACTIVE command
• allows the re-specification of the supports with another SUPPORT command that causes the old supports to be ignored. The SUPPORT specification must be such that the number of joint directions that are free to move (DOF or "releases") before the CHANGE must be greater than or equal to the number of "releases" after the CHANGE.
• the supports must be specified in the same order before and after the CHANGE command. To accomplish this when some cases have more supports than others do, you can enter unrestrained joints into the SUPPORT command list using FIXED BUT FX FY FZ MX MY MZ. It is best to put every joint that will be supported in any case into every SUPPORT list.
• CHANGE, if used, should be after PERFORM ANALYSIS and before the next set of SUPPORT or LOADS.
• Only active cases are solved after the CHANGE command.
• Analysis and CHANGE are required between primary cases for PERFORM CABLE ANALYSIS.
• Analysis and CHANGE are required after UBC cases if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

Example

Before CHANGE is specified:

1 PINNED
2 FIXED BUT FX MY MZ
3 FIXED BUT FX MX MY MZ

After CHANGE is specified:

1 PINNED
2 FIXED
3 FIXED BUT FX MZ

The CHANGE command is not necessary when only member properties are revised to perform a new analysis. This is typically the case in which the user has asked for a member selection and then uses the PERFORM ANALYSIS command to reanalyze the structure based on the new member properties.
Notes

a. If new load cases are specified after the CHANGE command such as in a structure where the INACTIVE MEMBER command is used, the user needs to define the total number of primary load cases using the SET NL option (see TR.5 Set Command Specification (on page 2413) and Example 4).

b. Multiple Analyses using the CHANGE command should not be performed if the input file contains load cases involving dynamic analysis or Moving Load Generation.

c. Section forces and moments, stress and other results for postprocessing will use the last entered data for supports and member properties regardless of what was used to compute the displacements, end forces and reactions. So beware of changing member properties and releases after a CHANGE command.

Related Links

- G.19 Multiple Analyses (on page 2401)
- A. To add a change command (on page 934)

TR.39 Load List Specification

This command allows specification of a set of active load cases. All load cases made active by this command remain active until a new load list is specified.

This command is used to activate the load cases listed in this command and, in a sense, deactivate all other load cases not listed in this command. In other words, the loads listed are used for printing output and in design for performing the specified calculations. When the PERFORM ANALYSIS command is used, the program internally uses all load cases, regardless of LOAD LIST command, except after a CHANGE command. In these two cases, the LOAD LIST command allows the program to perform analysis only on those loads in the list. If the LOAD LIST command is never used, the program will assume all load cases to be active.

General Format

```
LOAD LIST { load-list | ALL }
```

Example

```
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 3
PRINT SUPPORT REACTIONS
CHECK CODE ALL
```

In this example, member forces will be printed for all load cases, whereas loading 1 and 3 will be used for printing support reactions and code-checking of all members.

Notes

a. The LOAD LIST command may be used for multiple analyses situations when an analysis needs to be performed with a selected set of load cases only. All load cases are automatically active before a first CHANGE command is used.
b. After a CHANGE command has been used anywhere in the data, it is good practice to specify the LOAD LIST command after an ANALYSIS command and before the next command; otherwise only the last case analyzed may be used in the design.

c. Do not enter this command within the loads data (from the first Load command in an analysis set to the associated Analysis command).

d. Additionally, load cases can also be short-listed using load envelopes. This may be needed when operations such as steel design need to distinguish between service load combinations and factored load combination cases.

Related Links
- A To create a load list (on page 937)

TR.40 Load Envelope

Load Envelopes are a means for clustering a set of load cases under a single moniker (number). If one or more tasks have to be performed for a set of load cases, such as, serviceability checks under steel design for one set of load cases, strength checks under steel design for another set of cases, etc., this feature is convenient.

It is an alternative to the LOAD LIST command described in TR39 Load List Specification (on page 2836).

General Format

Load envelopes are defined within a definition block. Each envelope can be assigned an optional type to specify the qualitative nature of the load or load combination cases in the envelope definition.

```
DEFINE ENVELOPE

Load-list ENVELOPE i ( TYPE { STRESS | SERVICEABILITY | COLUMN | CONNECTION | STRENGTH } )

...

END DEFINE ENVELOPE
```

Where:

-  \( i \) = the load envelope number.

Two of the load envelope types — SERVICEABILITY and STRENGTH — have specific meaning from the standpoint of steel design to certain codes. See Using Load Envelopes in Designing Steel Members for Strength and Serviceability (on page 2838).

Example

The first line within the DEFINE ENVELOPE command means that load cases numbered 1 to 8 make up the CONNECTION type load envelope 1. Similarly load case 9 to 15 define the SERVICEABILITY type load envelope 2.

```
DEFINE ENVELOPE
1 TO 8 ENVELOPE 1 TYPE CONNECTION
9 TO 15 ENVELOPE 2 TYPE SERVICEABILITY
```
Load Lists for Envelopes

For operations and calculations which are going to be based on the load cases contained in the envelopes, the command

```
LOAD LIST ENVELOPE load-env-list
```

must be specified prior to those commands.

For example, to print out the support reactions corresponding to load envelope 1, the following commands should be provided in the input file

```
LOAD LIST ENV 1
PRINT SUPPORT REACTIONS
```

Using Load Envelopes in Designing Steel Members for Strength and Serviceability

Most design codes require two types of checks to be performed

a. deflection
b. strength (capacity)

In codes that are based on the strength design method —like the AISC 360-05, 360-10, and Eurocode 3— deflection checks are normally required to be done on the serviceability load cases and strength checks on a different set of load cases which are the factored load cases.

Hence, it is necessary to convey to the program which load cases are to be considered for the deflection checks, and which ones for the strength related checks.

Using Load Envelopes, you can convey to the program this information using the type keywords SERVICEABILITY and STRENGTH. Additionally, prior to the design command, you must also specify the command LOAD LIST ENV load-env-list.

```
DEFINE ENVELOPE
101 TO 110 ENVELOPE 1 TYPE SERVICEABILITY
201 to 225 ENVELOPE 2 TYPE STRENGTH
END DEFINE ENVELOPE

LOAD LIST ENVELOPE 1 2

PARAMETER 1
CODE AISC UNIFIED
METHOD LRFD
FYLD 46 MEMB 27 37 67 TO 89
DFF 240 ALL
DJ1 ... DJ2 ...
...
CHECK CODE ALL
```

Note that if only one type of check is required —say for example, strength only— then the separation of load cases into the different categories is not needed which means these envelopes do not have to be created. In that case, you would only specify:

```
DEFINE ENVELOPE
101 TO 110 ENVELOPE 1 TYPE STRENGTH
END DEFINE ENVELOPE
```
event, the simple LOAD LIST command described in TR.39 Load List Specification (on page 2836) will suffice instead of LOAD LIST ENV load-env-list.

Related Links
- M. To create a load envelope (on page 877)
- M. To create a load envelope (on page 877)
- G.18 Member End Forces (on page 2395)
- G.18.1.1 Member Forces at Intermediate Sections (on page 2400)
- G.18.1.3 Member Stresses at Specified Sections (on page 2400)

TR.41 Section Specification

This command is used to specify sections along the length of frame member for which forces and moments are required.

This command specifies the sections, in terms of fractional member lengths, at which the forces and moments are considered for further processing.

General Format

```
SECTION f1 (f2) (f3) { MEMBER memb-list | (ALL) }
```

Where:

\(f_1, f_2, f_3\) Section (in terms of the fraction of the member length) provided for the members. The maximum number of sections is three, and only values between 0.0 and 1.0 will be considered. In other words, no more than three intermediate sections are permissible per SECTION command.

Example

```
SECTION 0.17 0.48 0.72 MEMB 1 2
SECTION 0.25 0.75 MEMB 3 TO 7
SECTION 0.6 MEMB 8
```

In the above example, first, section locations of 0.17, 0.48, and 0.72 are set for members 1 and 2. In the next SECTION command, sections 0.25 and 0.75 are set for members 3 to 7. In the third SECTION command, member 8 has its section specified at 0.6. The remainder of the members will have no sections provided for them. As mentioned earlier, no more than three intermediate sections are allowed per SECTION command. However, if more than three intermediate sections are desired, they can be examined by repeating the SECTION command after completing the required calculations. The following example will clarify.

Example

```
SECTION 0.2 0.4 0.5 ALL
PRINT SECTION FORCES
SECTION 0.6 0.75 0.9 ALL
PRINT SECTION FORCES
```
In this example, forces at three intermediate sections (namely 0.2, 0.4 and 0.5) are printed. Then forces at an additional 3 sections (namely 0.6, 0.75 and 0.9) are printed. This gives the user the ability to obtain section forces at more than three intermediate sections.

Notes

a. The SECTION command just specifies the sections. Use the PRINT SECTION FORCES command after this command to print out the forces and moments at the specified sections.
b. This is a secondary analysis command. The analysis must be performed before this command may be used.
c. To obtain values at member ends (START and END), use the PRINT MEMBER FORCES command.
d. If this command is specified before a steel design operation and if the BEAM parameter is set to zero, the section locations specified by this command will also be designed for, in addition to the BEAM ends. This can be used to specify three locations along the beam for design when the default sections are not sufficient. See Note 9 in AISC 360 Design Parameters (on page 1378).

Related Links

- G.18 Member End Forces (on page 2395)
- G.18.1.1 Member Forces at Intermediate Sections (on page 2400)
- G.18.1.3 Member Stresses at Specified Sections (on page 2400)
- G.18.1 Secondary Analysis (on page 2400)
- G.18.1.2 Member Displacements at Intermediate Sections (on page 2400)

TR.42 Print Specifications

This command is used to direct the program to print various modeling information and analysis results. STAAD.Pro offers a number of versatile print commands that can be used to customize the output.

General Format

Data Related Print Commands

```
PRINT { PROBLEM STATISTICS | JOINT COORDINATES | MEMBER PROPERTIES | ELEMENT INFORMATION (SOLID) | MATERIAL PROPERTIES | SUPPORT INFORMATION | ALL } { (ALL) | LIST item/joint/member-list }
```

Print Location of CG

```
PRINT CG (_group_name)
```

Print Analysis Results

```
PRINT { (JOINT) DISPLACEMENTS | MEMBER FORCES (GLOBAL) | ANALYSIS RESULTS | MEMBER SECTION FORCES | MEMBER STRESSES | ELEMENT (JOINT) STRESSES (AT f1 f2) | ELEMENT FORCES | ELEMENT (JOINT) STRESSES SOLID | MODE SHAPES } list-spec
```

```
list-spec = { (ALL) | LIST joint/member/elements-list }
```

Print Support Reactions

```
PRINT SUPPORT REACTIONS
```
Print Story Drifts or Stiffness
PRINT  STORY  DRIFT  (f_s)
PRINT  STORY  STIFFNESS

Print Location of Center of Rigidity at each Floor
PRINT  DIAPHRAGM  CR

Print the deflected (sag) shape of a cable after an advanced cable analysis:
PRINT  CABLE  SAG

Description

The list of items is not applicable for PRINT  ANALYSIS  RESULTS  and PRINT  MODE  SHAPES commands.

The PRINT  JOINT  COORDINATES command prints all interpreted coordinates of joints.

The PRINT  MEMBER  INFORMATION command prints all member information, including member length, member incidences, beta angles, whether or not a member is a truss member and the member release conditions at start and end of the member (1 = released, 0 = not released).

The PRINT  ELEMENT  INFORMATION command prints all incident joints, element thicknesses, and Poisson ratios for Plate/Shell elements. The PRINT  ELEMENT  INFORMATION  SOLID command prints similar information for Solid elements.

The PRINT  MEMBER  PROPERTIES command prints all member properties including cross sectional area, moments of inertia, and section moduli in both axes. Units for the properties are always INCH or CM (depending on FPS or METRIC) regardless of the unit specified in UNIT command.

The following designation is used for member property names:

- **AX**  Cross section area
- **AY**  Area used to adjust shear/bending stiffness in local Y axis to account for pure shear in addition to the classical bending stiffness
- **AZ**  Area used to adjust shear/bending stiffness in local Z axis to account for pure shear in addition to the classical bending stiffness.
- **IZ**  Moment of Inertia about the local Z-axis
- **IY**  Moment of Inertia about the local Y-axis
- **IX**  Torsional constant
- **SY**  Smallest section modulus about the local Y-axis
- **SZ**  Smallest section modulus about the local Z-axis

The PRINT  MATERIAL  PROPERTIES command prints all material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members. This command is available for members only. G may be listed as zero if command is before load cases and Poisson ratio was entered but G was not entered.

The PRINT  SUPPORT  INFORMATION command prints all support information regarding their fixity, releases and spring constant values, if any. The LIST option is not available for this command.
The PRINT ALL command is equivalent to last five print commands combined. This command prints joint coordinates, member information, member properties, material properties and support information, in that order.

The PRINT CG command prints out the coordinates of the center of gravity and the total weight of the structure or of a single group of member/elements. If the CG of a portion of the structure is desired, the members and elements of that portion must be assigned using a group name (see TR.16 Entities as Single Objects (on page 2440) for details on using group-names). Only the selfweight of the structure is used to calculate the C.G. User defined joint loads, member loads etc. are not considered in the calculation of C.G.

The PRINT (JOINT) DISPLACEMENTS command prints joint displacements in a tabulated form. The displacements for all six directions will be printed for all specified load cases. The length unit for the displacements is always INCH or CM (depending on FPS or METRIC unit) regardless of the unit specified in UNIT command.

The PRINT (MEMBER) FORCES command prints member forces (i.e., Axial force (AXIAL), Shear force in local Y and Z axes (SHEAR-Y and SHEAR-Z), Torsional Moment (TORSION), Moments about local Y and Z axes (MOM-Y and MOM-Z)) in a tabulated form for the listed members, for all specified load cases. The GLOBAL option will output forces in the global coordinate system rather than the member local coordinate system for each member.

The PRINT SUPPORT REACTIONS command prints global support reactions in a tabulated form, by support, for all specified load cases. Use LIST option for selected joints.

The PRINT ANALYSIS RESULTS command is equivalent to the above three commands combined. With this command, the joint displacements, support reactions and member forces, in that order, are printed.

The PRINT (MEMBER) SECTION FORCES command prints axial force, shear forces, & bending moments at the intermediate sections specified with a previously input SECTION command. The printing is done in a tabulated form for all specified cases for the first requested member, then for the next member, etc.

Note: An asterisk following a critical load case number within PRINT JOINT DISPLACEMENT, PRINT MEMBER FORCES, PRINT SECTION FORCES, PRINT MEMBER STRESS, or PRINT SUPPORT REACTION outputs indicates that this load case is a generated load combination. See TR.35 Load Combination Specification (on page 2791) for additional information.

The PRINT (MEMBER) STRESSES command tabulates member stresses at the start joint, end joint and all specified intermediate sections. These stresses include axial (i.e., axial force over the area), bending-y (i.e., moment-y over section modulus in local y-axis), bending-z (i.e., moment-z over section modulus in local z-axis), shear stresses in both local y and z directions (shear flow, q, over the shear area), and combined (absolute combination of axial, bending-y and bending-z) stresses.

- For PRISMATIC sections, if AY and/or AZ is not provided, the full cross-sectional area (AX) will be used.
- For TAPERED sections, the values of AY and AZ are those for the location where the stress is printed. Hence at the location 0.0, the AY and AZ are based on the dimensions of the member at the start node.

The PRINT ELEMENT STRESSES command must be used to print plate stresses (SX, SY, SXY, SQX, SQY), moments per unit width (MX, MY, MXY) and principal stresses (SMAX, SMIN, TMAX) for plate/shell elements. Typically, the stresses and moments per unit width at the centroid will be printed. The Von Mises stresses (VONT, VONB) as well as the angle (ANGLE) defining the orientation of the principal planes are also printed.

The variables that appear in the output are the following. Refer to G.5.1 Plate and Shell Elements (on page 2308) for more information regarding these variables.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SQX</td>
<td>Shear stress on the local X face in the Z direction</td>
</tr>
<tr>
<td>SQY</td>
<td>Shear stress on the local Y face in the Z direction</td>
</tr>
</tbody>
</table>
Technical Reference of STAAD Commands

TR.42 Print Specifications

MX  Moment per unit width about the local X face
MY  Moment per unit width about the local Y face
MXY Torsional Moment per unit width in the local X-Y plane
SX  Axial stress in the local X direction
SY  Axial stress in the local Y direction
SXY Shear stress in the local XY plane
VONT Von Mises stress on the top surface of the element
VONB Von Mises stress on the bottom surface of the element
TrescaT Tresca stress on the top surface of the element
TrescaB Tresca stress on the bottom surface of the element
SMAX Maximum in-plane Principal stress
SMIN Minimum in-plane Principal stress
TMAX Maximum in-plane Shear stress
ANGLE Angle which determines direction of maximum principal stress with respect to local X axis

Note: If the JOINT option is used, forces and moments at the nodal points are also printed out in addition to the centroid of the element.

The AT option may be used to print element forces at any specified point within the element. The AT option must be accompanied by \( f_1 \) and \( f_2 \). \( f_1 \) and \( f_2 \) are local X and Y coordinates (in current units) of the point where the stresses and moments are required. For detailed description of the local coordinate system of the elements, refer to G.5 Finite Element Information (on page 2308).

The PRINT ELEMENT FORCES command enables printing of plate “corner forces” \( F_p = K_p \cdot D_p \) in global axis directions.

The PRINT ELEMENT (JOINT) STRESS SOLID command enables printing of stresses at the center of the SOLID elements. The variables that appear in the output are the following.

Normal Stresses \( S_{XX}, S_{YY} \) and \( S_{ZZ} \)
Shear Stresses \( S_{XY}, S_{YZ}, S_{ZX} \)
Principal Stresses \( S_1, S_2, S_3 \)
Von Mises Stresses \( S_E \)
Direction cosines Six direction cosines are printed following the expression DC, corresponding to the first two principal stress directions.

Note: The JOINT option will print out the stresses at the nodes of the solid elements.

The PRINT MODE SHAPES command prints the relative joint motions of each of the modes that were calculated. The maximum motion is arbitrary and has no significance. Dynamic analysis will scale and combine the mode shapes to achieve the final dynamic results.
Example

PERFORM ANALYSIS
PRINT ELEMENT JOINT STRESS
PRINT ELEMENT STRESS AT 0.5 0.5 LIST 1 TO 10
PRINT SUPPORT REACTIONS
PRINT JOINT DISPLACEMENTS LIST 1 TO 50
PRINT MEMBER FORCES LIST 101 TO 124

Example

Printing the Center of Gravity (CG)

PRINT CG
PRINT CG_RAFTERBEAMS
PRINT CG_RIDGEBEAMS

Notes

1. The output generated by these commands is based on the current unit system. The user may wish to verify the current unit system and change it if necessary.
2. Results may be printed for all joints/members/elements or based on a specified list.

Printing the Story Drift and Stiffness

The PRINT STORY DRIFT command may be used to obtain a print-out of the average lateral displacement of all joints at each horizontal level along the height of the structure.

The procedure used in STAAD.Pro for calculating story drift is independent of any code. For example, the story drift determination as explained in section 12.8.6 of the ASCE 7-05 code is not implemented in STAAD.Pro.

The method implemented in STAAD.Pro involves:

a. Find all the distinct Y coordinates in the model. Those are what STAAD.Pro calls as stories.
b. For each of those distinct stories, find all the nodes at that story elevation.
c. For each story, find the average displacement along the horizontal directions (X and Z) by adding up corresponding displacement for all the nodes at that story, and dividing by the number of nodes for that story. Thus, even if there is only a single node representing a story, a drift is calculated for that story too.

In STAAD.Pro if PRINT STORY DRIFT command is issued, the program prints the average of horizontal displacements of all the joints present at the particular floor level.

However, to check inter-story drift, the following commands needs to be issued after the PERFORM ANALYSIS command.

LOAD LIST i
PRINT STORY DRIFT f3

Where:
The primary load number for which inter-story drift check is required

\( f_d = \text{The allowable drift factor, as per the code provision} \)

The program will calculate the relative horizontal displacement between two adjacent floors. This calculated value is checked against the allowable limit. The result is reported as either "PASS" or "FAIL" in the output.

**Note:**

There is only one exception to this format. For IS 1893: 2002 static seismic load case, even if this factor is not provided the program internally checks if the loading is IS 1893 static seismic loading or not. If yes, it automatically calculates the inter-story drift and checks against the code provisions.

For dynamic IS 1893: 2002 response spectrum analysis the above format for inter-story drift check does not hold true. The reason is that in response spectrum analysis the joint displacements represent the maximum magnitude of the response quantity that is likely to occur during seismic loading. Any response quantity like story drift should be calculated from actual displacements of each mode considered during analysis. The inter-story drift from each mode is combined using modal combination to get the maximum magnitude of this response quantity. In order to compute story drift for IS 1893 response spectrum the load case command format described in TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2721) must be used.

The PRINT STORY STIFFNESS command may be used to include the calculated lateral stiffness of each story used in determining the drift. In STAAD.Pro lateral stiffness is calculated only when the floor is modeled as rigid floor diaphragm since it functions as transferring story shears and torsional moments to lateral force-resisting members during earthquake.

**Printing the Center of Rigidity & Center of Mass**

The PRINT DIAPHRAGM CR command may be used to obtain a print-out of the center of rigidity and center of mass at each rigid diaphragm in the model. The lateral force at each floor, as generated by earthquake and wind loading, acts at the center of rigidity of each floor which is modeled as rigid floor diaphragm. The center of mass of each floor is defined as the mean location of the mass system of each floor. The mass of the floor is assumed to be concentrated at this point when the floor is modeled as rigid diaphragm. The distance between these two is the lever arm for the natural torsion moment for seismic loads when that option is used.

**Related Links**
- A. To specify post-analysis print commands (on page 939)
- A. To specify pre-analysis commands (on page 936)
- A. To output the center of rigidity (on page 940)
- A. To check for soft stories and seismic code irregularities (on page 937)
- A. To check for inter-story drift (on page 941)
- G.22 Miscellaneous Facilities (on page 2402)
- G.18 Member End Forces (on page 2395)
- G.18.1.1 Member Forces at Intermediate Sections (on page 2400)
- G.18.1.3 Member Stresses at Specified Sections (on page 2400)
- G.18.1 Secondary Analysis (on page 2400)
- G.18.1.2 Member Displacements at Intermediate Sections (on page 2400)
- G.5.3 Surface Elements (Deprecated) (on page 2322)
TR.43 Stress/Force Output Printing for Surface Entities

Default locations for stress/force output, design, and design output for surface elements are set as follows:

- **SURFACE DIVISION X xd**
- **SURFACE DIVISION Y yd**

Where:

- \( xd \) = number of divisions along X axis,
- \( yd \) = number of divisions along Y axis.

\( xd \) and \( yd \) represent default numbers of divisions for each edge of the surface where output is requested. The output is provided for sections located between division segments. For example, if the number of divisions \( = 2 \), then the output will be produced for only one section (at the center of the edge).

Values of internal forces may be printed out for any user-defined section of a wall modeled using the Surface element.

**General Format**

```
PRINT SURFACE FORCE (ALONG \{X | Y\}) (AT a) (BETWEEN d1 d2) LIST s1 s2 ... si
```

Where:

- \( a \) = distance along the local axis from start of the member to the full cross-section of the wall,
- \( d1 d2 \) = coordinates in the direction orthogonal to \( x \), delineating a fragment of the full cross-section for which the output is desired.
- \( s1 s2 ... si \) = list of surfaces for output generation

ALONG specifies local axis of the surface element

**Notes**

- **a.** If the keyword ALONG is omitted, direction Y (default) is assumed.
- **b.** If command AT is omitted, output is provided for all sections along the specified (or default) edge. The number of sections will be determined from the SURFACE DIVISION X or SURFACE DIVISION Y input values.
- **c.** If the BETWEEN keyword is omitted, the output is generated based on full cross-section width.

**Related Links**

- [G.18.1.4 Force Envelopes](on page 2401)
- [G.5.3 Surface Elements (Deprecated)](on page 2322)

TR.44 Printing Section Displacements for Members

This command is used to calculate and print displacements at sections (intermediate points) of frame members. This provides the user with deflection data between the joints.
General Format

PRINT SECTION (MAX) DISPLACEMENTS (NSEC i) (SAVE a) \{ NOPRINT | ALL | LIST memb-list \}

Where:

i = number of sections to be taken. Defaults to 12 if NSEC is not used and also if SAVE is used (max=24, min=2).
a = File name where displacement values can be stored and used by the STAADPL graphics program. If the NOPRINT command is used in conjunction with the SAVE command, the program writes the data to file only and does not print them in the output.
This option is not necessary in STAAD.Pro.

Description

This command prints displacements at intermediate points between two joints of a member. These displacements are in global coordinate directions (see figure). If the MAX command is used, the program prints only the maximum local displacements among all load cases.

Example

PRINT SECTION DISPL SAVE
PRINT SECTION MAX DISP

SECTION DISPLACEMENTS are measured in GLOBAL COORDINATES. The values are measured from the original (undeflected) position to the deflected position. See figure above.

The maximum local displacement is also printed. First, the location is determined and then the value is measured from this location to the line joining start and end joints of the deflected member.
Notes

a. The section displacement values are available in Global Coordinates. The undeflected position is used as the datum for calculating the deflections.
b. This is a secondary analysis command. An analysis must be performed before this command may be used.

Related Links

- *G.21 Printing Facilities* (on page 2402)
- *G.18.1 Secondary Analysis* (on page 2400)
- *G.18.1.2 Member Displacements at Intermediate Sections* (on page 2400)

TR.45 Printing the Force Envelope

This command is used to calculate and print force/moment envelopes for frame members. This command is not available for finite elements.

General Format

```plaintext
PRINT { FORCE | MAXFORCE } ENVELOPE (NSECTION i) list-spec
list-spec = { LIST memb-list | (ALL) }
```

Description

Where:

**NSECTION i** the number of equally spaced sections to be considered in printing maximum and minimum force envelopes. i is an integer up to 96, though not recommended to be less than 12. If the NSECTION i command is omitted, i defaults to 12.
The MAXFORCE command produces maximum/minimum force values only of all sections, whereas the FORCE command prints maximum/minimum force values at every section as well as the max/min force values of all sections. The force components include FY, MZ, FZ, and MY. The SECTION command (as described in TR.40 Load Envelope (on page 2837)) does not define the number of sections for force envelopes. For the sign convention of force values, refer to G.18 Member End Forces (on page 2395).

**Note:** This is a secondary analysis command and should be used after analysis specification.

**Example**

```
PRINT FORCE ENV
PRINT MAXF ENV NSE 24
PRINT FORCE ENV NSE 24 LIST 3 TO 15
```

**Related Links**

- G.21 Printing Facilities (on page 2402)
- G.18.1.4 Force Envelopes (on page 2401)
- G.18.1 Secondary Analysis (on page 2400)
- G.18.1.2 Member Displacements at Intermediate Sections (on page 2400)

**TR.46 Post Analysis Printer Plot Specifications**

This command has been discontinued in STAAD.Pro. Please use the facilities of the Graphical User Interface (GUI) for screen and hard copy graphics.

**Related Links**

- G.21 Printing Facilities (on page 2402)

**TR.47 Size Specification**

This command provides an estimate for required section properties for a frame member based on certain analysis results and user requirements.

**General Format**

```
SIZE *{ WIDTH f1 | DEFLECTION f2 | LENGTH f3 | BSTRESS f4 | SSTRESS f5 } { MEMBER memb-list | ALL }
```

Where:

- \( f_1 \) = allowable width
- \( f_2 \) = Maximum allowable (Length/Maxm. local deflection) ratio
- \( f_3 \) = Length for calculating the above ratio. The default value is the actual member length.
- \( f_4 \) = Maximum allowable bending stress.
- \( f_5 \) = Maximum allowable shear stress.

The values must be provided in the current unit system.
Description

This command may be used to calculate required section properties for a member based on analysis results and user specified criteria. The user specified criteria may include Member width, Allowable (Length/Maxm. Deflection) Ratio, Maxm. allowable bending stress and Maximum allowable shear stress. Any number of these criteria may be used simultaneously. The output includes required Section Modulus (about major axis), required Shear Area (for shear parallel to minor axis), Maxm. moment capacity (about major axis), Maxm. shear capacity (for shear parallel to minor axis) and Maxm. (Length/local maxm. deflection) ratio.

Example

SIZE WID 12 DEFL 300 LEN 240 BSTR 36 ALL
SIZE DEFL 450 BSTR 42 MEMB 16 TO 25

Note: It may be noted that sizing will be based on only the criteria specified by the user in the relevant SIZE command.

In the first example above, sizing will be based on user specified member width of 12, Length/Deflection ratio of 300 (where Length= 240) and max. allowable bending stress of 36.

In the second example, sizing will be based on Length/Deflection ratio of 450 (where Length= actual member length) and max. allowable bending stress of 42.

Note: This is a post-analysis facility and must be used after the analysis specifications.

TR.48 Steel and Aluminum Design Specifications

This section describes the specifications necessary for structural steel & aluminum design.

The specific details of the implementation of these codes may be found in:

- D. Available Steel Design Codes (on page 944)
- D. Available Aluminum Design Codes (on page 1362)

Additional related information:

- TR.48.1 Parameter Specifications (on page 2851) discusses specification of the parameters that may be used to control the design.
- TR.49 Code Checking Specification (on page 2852) and TR.49.1 Member Selection Specification (on page 2853) describe the CODE CHECKING and MEMBER SELECTION options respectively.
- TR.49.2 Member Selection by Optimization (on page 2854) discusses member selection by optimization.

Shortlisting Load Cases Used for Design

There are two options available to instruct STAAD.Pro to use only specific load cases for design:

1. The LOAD LIST command.
2. The LOAD LIST ENVELOPE command, which can only be used if load envelopes have been created (using the DEFINE ENVELOPE command). Once envelopes have been created, this command must be used or the design
will be based on all active load cases or those specified in a LOAD LIST command, rather than only the envelopes.

Related Links
- Design Commands dialog (on page 3084)
- Steel Design - Whole Structure dialog (on page 3081)
- D. To design steel members in groups (on page 951)
- D. To specify steel design commands (on page 952)

TR.48.1 Parameter Specifications

This set of commands may be used to specify the parameters required for steel and aluminum design.

General Format

PARAMETER

<table>
<thead>
<tr>
<th>CODE</th>
<th>design-code</th>
</tr>
</thead>
</table>

{ parameter-name f1 | PROFILE a1 (a2 a3) } { MEMBER memb-list | ALL | member-group-name | deck-name }

where:

- **CODE design-code**
  The CODE parameter let you choose the type of steel or aluminum code to be checked for design. The default steel code depends on the country of distribution.

<table>
<thead>
<tr>
<th>design-code</th>
</tr>
</thead>
<tbody>
<tr>
<td>AASHTO</td>
</tr>
<tr>
<td>AISC</td>
</tr>
<tr>
<td>AISI</td>
</tr>
<tr>
<td>ALUMINUM</td>
</tr>
<tr>
<td>S136</td>
</tr>
<tr>
<td>AUSTRALIAN</td>
</tr>
<tr>
<td>BRITISH</td>
</tr>
<tr>
<td>CANADIAN</td>
</tr>
<tr>
<td>FRENCH</td>
</tr>
<tr>
<td>GERMAN</td>
</tr>
<tr>
<td>INDIA</td>
</tr>
<tr>
<td>JAPAN</td>
</tr>
<tr>
<td>LRFD</td>
</tr>
<tr>
<td>NORWAY</td>
</tr>
<tr>
<td>BS5400</td>
</tr>
<tr>
<td>RUSSIA</td>
</tr>
<tr>
<td>LRFD2</td>
</tr>
<tr>
<td>API</td>
</tr>
<tr>
<td>TIMBER</td>
</tr>
<tr>
<td>SPANISH</td>
</tr>
<tr>
<td>CHINA</td>
</tr>
<tr>
<td>EN1993</td>
</tr>
<tr>
<td>ASCE</td>
</tr>
<tr>
<td>DUTCH</td>
</tr>
<tr>
<td>NPD</td>
</tr>
<tr>
<td>DANISH</td>
</tr>
<tr>
<td>BSK94</td>
</tr>
<tr>
<td>FINNISH</td>
</tr>
<tr>
<td>IS801</td>
</tr>
<tr>
<td>IS802</td>
</tr>
<tr>
<td>MEXICAN</td>
</tr>
<tr>
<td>BAS59520 1990</td>
</tr>
<tr>
<td>NZS3404 1997</td>
</tr>
</tbody>
</table>

- **parameter-name f1**
  refers to the Parameter Names listed in the parameter table contained in the Steel Design and Aluminum Design sections. f1 refers to the value of the parameter. Not all parameters require values.
The details of the parameters available for specific codes may be found in D. Design Codes (on page 1366).

You can control the design through specification of proper parameters.

**PROFILE a1 (a2 a3)**

The PROFILE parameter is available for only a limited number of codes, such as AISC ASD and AISC LRFD. You can specify up to three profiles (a1, a2 and a3). Profile is the first three letters of a section name from its steel table, like, W8X, W12, C10, L20 etc. The PROFILE parameter-name is used only for member selection where members are selected from each of those profile names. The PROFILE for a T-section is the corresponding W-shape. Also, the shape specified under PROFILE has to be the same as that specified initially under MEMBER PROPERTIES.

**memb-list**

A list of member numbers

**member-group-name**

Please see TR.16 Entities as Single Objects (on page 2440) for the definition of member group names.

**deck-name**

Deck names are explained in TR.20.7 Composite Decks (on page 2473).

---

**Example**

```
PARAMETERS
CODE AISC
KY 1.5 MEMB 3 7 TO 11
NSF 0.75 ALL
PROFILE W12 W14 MEMB 1 2 23
RATIO 0.9 ALL
```

**Notes**

a. All unit sensitive values should be in the current unit system.

b. For default values of the parameters, refer to the appropriate parameter table.

**Related Links**

- D. To specify steel design code and parameters (on page 951)
- D. To specify aluminum design code and parameters (on page 1363)

---

**TR.49 Code Checking Specification**

Used to perform the CODE CHECKING operation for steel and aluminum members.

This command checks the specified members against the specification of the desired code. Refer to D. Steel Design (on page 944) for detailed information.

**General Format**

```
CHECK CODE { MEMBER memb-list | ALL | member-group-name | deck-name | PMEMB pmember-list }
```
Example

CHECK CODE MEMB 22 TO 35
CHECK CODE PMEMB 1 3
CHECK CODE _BEAMS
CHECK CODE ALL
CHECK CODE PMEMB ALL

Notes

a. The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in TR.48.1 Parameter Specifications (on page 2851) for more information on the TRACK parameter.

b. Member group names and deck names are explained in TR.16 Entities as Single Objects (on page 2440) and TR.20.7 Composite Decks (on page 2473) respectively.

c. The PMEMB list is explained in TR.16.2 Physical Members (on page 2443).

Related Links

- D. To specify steel design commands (on page 952)
- D. To specify aluminum design commands (on page 1363)
- D1.B.1.3 Code Checking (on page 1418)
- D1.C.7 Code Checking and Member Selection (on page 1458)

TR.49.1 Member Selection Specification

This command may be used to perform the MEMBER SELECTION operation.

This command instructs STAAD.Pro to select specified members based on the parameter value restrictions and specified code. The selection is done using the results from the most recent analysis and iterating on sections until a least weight size is obtained. Refer to D. Steel Design (on page 944) for more details.

General Format

SELECT { MEMBER memb-list | ALL | member-group-name | deck-name | PMEMB pmember-list }

It is important that the keywords MEMBER, ALL, or PMEMB be provided. Thus, the keyword SELECT by itself is not sufficient.

Example

SELECT MEMB 22 TO 35
SELECT PMEMB 1 3
SELECT _COLUMNS
SELECT ALL
SELECT PMEMB ALL
Notes

a. The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in TR.48.1 Parameter Specifications (on page 2851) for more information on the TRACK parameter.

b. Member selection can be done only after an analysis has been performed. Consequently, the command to perform the analysis has to be specified before the SELECT MEMBER command can be specified.

c. This command does not cause the program to re-analyze for results based on the selected member sizes. However, to maintain compatibility of analysis results with the final member sizes, you should enter a subsequent PERFORM ANALYSIS command. Otherwise the post processor will display the prior results with the revised member sizes.

d. Member group names and deck names are explained in TR.16 Entities as Single Objects (on page 2440) and TR.20.7 Composite Decks (on page 2473) respectively.

e. The PMEMB list is explained in TR.16.2 Physical Members (on page 2443).

Related Links

- D. To specify steel design commands (on page 952)
- D. To specify aluminum design commands (on page 1363)
- D. To specify aluminum design commands (on page 1363)
- D1.B.1.4 Member Selection (on page 1419)
- D1.B.1.3 Code Checking (on page 1418)
- D1.C.7 Code Checking and Member Selection (on page 1458)

TR.49.2 Member Selection by Optimization

This command performs member selection using an optimization technique based on multiple analysis/design iterations.

The program selects all members based on an optimization technique. This method performs 2 analyses as well as iteration of sizes to reduce the overall structure weight. This command should be used with caution since it will require longer processing time.

General Format

SELECT OPTIMIZED

Notes

a. The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in TR.48.1 Parameter Specifications (on page 2851) for more information on the TRACK parameter.

b. This command will perform one additional analysis and design cycle and therefore may be time consuming. Steps taken are: CHECK CODE ALL; then modify ratios; then SELECT ALL; then PERFORM ANALYSIS; then SELECT ALL. See TR.50 Group Specification (on page 2855) for other options used with this command. You may want to repeat this command for further optimization.

c. This command does not cause the program to re-analyze for results based on the selected member sizes. However, to maintain compatibility of analysis results with the final member sizes, you should enter a subsequent PERFORM ANALYSIS command. Otherwise the post processor will display the prior results with the revised member sizes.
Related Links

- **D. To specify steel design commands** (on page 952)
- **D. To specify aluminum design commands** (on page 1363)
- **D1.B.1.3 Code Checking** (on page 1418)
- **D1.C.7 Code Checking and Member Selection** (on page 1458)

TR.50 Group Specification

This command may be used to group members together for analysis and steel design.

General Format

\[(\text{FIXED GROUP})\]

\[
\text{GROUP (prop-spec) MEMBER memb-list (SAME AS } i1)\]

\[
\text{prop-spec} = \{ \text{AX} | \text{SY} | \text{SZ} \}
\]

Where:

\text{SAME AS } i1 \quad \text{member number used in the SAME AS command. If provided, the program will group the members based on the properties of } i1.

Description

This command enables the program to group specified members together for analysis based on their largest property specification.

When \text{FIXED GROUP} is omitted, the \text{GROUP} command is usually entered after the member selection command, and the selected members will be grouped immediately and the new member properties will be used in any further operations. After the grouping is completed, the \text{GROUP} commands are discarded and will not be used again. Further grouping will be done only if a new \text{GROUP} command is encountered later.

If the \text{FIXED GROUP} option precedes the group data, the specified grouping will be retained in memory by the program and will be used in subsequent \text{SELECT} commands. No grouping will occur unless a \text{SELECT (MEMBER or ALL or OPTIMIZED)} command is performed. However, grouping will be performed with every subsequent \text{SELECT} command.

---

**Example 1**

\[
\text{SELECT ALL} \\
\text{GROUP SZ MEMB 1 3 7 TO 12 15} \\
\text{GROUP MEMB 17 TO 23 27 SAME AS 30}
\]

In this example, the members 1, 3, 7 to 12, and 15 are assigned the same properties based on which of these members has the largest section modulus. Members 17 to 23 and 27 are assigned the same properties as member 30, regardless of whether member 30 has a smaller or larger cross-sectional area. AX is the default property upon which grouping is based.
Example 2

FIXED GROUP
GROUP MEMB 1 TO 5
SELECT OPTIMIZED

In the above example, the usage of the FIXED GROUP command is illustrated. In this example, the SELECT OPTIMIZED command involves the six-stage process of

1. CHECK CODE ALL followed by modification of RATIO
2. SELECT ALL
3. GROUPING MEMBERS 1 TO 5
4. PERFORM ANALYSIS
5. SELECT ALL
6. GROUPING MEMBERS 1 TO 5

The FIXED GROUP command (and the GROUP commands that follow it) is required for execution of steps 3 and 6 in the cycle. You may want to repeat this data for further optimization.

Notes

The FIXED GROUP + GROUP commands are typically entered before the member selection for further analysis and design. This facility may be effectively utilized to develop a practically oriented design where several members need to be of the same size.

All the members in a list for a specific GROUP command should have the same cross section type. Thus, if the command reads

GROUP MEMB 1 TO 10

and member 3 is a W shape and member 7 is a Channel, grouping will not be done. The 10 members must be either all W shapes or all channels.

Also, refer to Note 'C' in TR.49.1 Member Selection Specification (on page 2853).

Related Links

• D. To design steel members in groups (on page 951)

TR.51 Steel and Aluminum Take Off Specification

This command may be used to obtain a summary of all steel sections and aluminum sections being used along with their lengths and weights (quantity estimates).

This command provides a listing of the different steel and aluminum table sections used in the members selected. For each section name, the total length and total weight of all members which have been assigned that section will be listed in a tabular form. This can be helpful in estimating steel and aluminum quantities.

General Format

{ STEEL | ALUMINUM } (MEMBER) TAKE (OFF) ( { LIST memb-list | LIST membergroupname | ALL } )
If the MEMBER option is specified, the length and weight of each member and the section name it is assigned will be reported.

Example

STEEL TAKE OFF LIST 71 TO 85
ALUMINUM TAKE OFF LIST PLGNO3

Related Links
- D. To generate aluminum take off (on page 1364)
- D. To generate steel take off (on page 953)

TR.52 Timber Design Specifications

This section describes the specifications required for timber design.

Related Links
- D. Available Timber Design Codes (on page 1364)
- Timber Design - Whole Structure dialog (on page 3088)
- Design Parameters dialog (on page 3083)
- D. To specify timber design code and parameters (on page 1365)
- D1.G.1 Design Operations (on page 1527)
- D1.G.2 Allowable Stress per AITC Code (on page 1528)
- D1.G.3 Input Specification (on page 1530)

TR.52.1 Timber Design Parameter Specifications

This set of commands may be used for specification of parameters for timber design.

General Format

PARAMETER

CODE  design-code

parameter-name f1 { MEMBER member-list | ALL }

Where:

parameter-name f1 = the name and corresponding value of the parameter.

<table>
<thead>
<tr>
<th>Code</th>
<th>design-code Value</th>
<th>Parameters List</th>
</tr>
</thead>
</table>

or

AITC
## Code Design Specifications

<table>
<thead>
<tr>
<th>Code</th>
<th>design-code Value</th>
<th>Parameters List</th>
</tr>
</thead>
<tbody>
<tr>
<td>AITC 1984</td>
<td>AITC 1984 or TIMBER</td>
<td>D1.G.5 Design Parameters (on page 1533)</td>
</tr>
<tr>
<td>CAN/CSA -086-01</td>
<td>TIMBER CANADIAN</td>
<td>D4.D.6 Design Parameters (on page 1724)</td>
</tr>
<tr>
<td>EC 5: Part 1-1</td>
<td>TIMBER EC5</td>
<td>D5.E.3 Design Parameters (on page 1863)</td>
</tr>
</tbody>
</table>

**Note:** All values must be provided in the current unit system.

### Related Links
- D1.G.1 Design Operations (on page 1527)
- D1.G.2 Allowable Stress per AITC Code (on page 1528)
- D1.G.3 Input Specification (on page 1530)

### TR.52.2 Code Checking Specification

This command performs code checking operation on specified members based on the American Institute of Timber Construction (AITC), Canadian Standards Agency (CSA), or Eurocode (EC5) codes.

The results of the code checking are summarized in a tabular format. Examples and detailed explanations of the tabular format are available in D1.G. American Codes - Timber Design per AITC Code (on page 1526).

#### General Format

CHECK CODE { MEMBER member-list | ALL }

**Note:** For the EC5 code, the output of this command may be controlled by the TRACK parameter.

### Related Links
- D1.G.6 Member Design Capabilities (on page 1538)

### TR.52.3 Member Selection Specification

This command performs member selection operation on specified members based on the American Institute of Timber Construction (AITC 1984) code.

This command may be used to perform member selection according to the AITC 1984 code. The selection is based on the results of the latest analysis and iterations are performed until the least weight member satisfying all the applicable code requirements is obtained. Parameters may be used to control the design and the results are available in a tabular format. Detailed explanations of the selection process and the output are available in D1.G. American Codes - Timber Design per AITC Code (on page 1526).
General Format

SELECT { MEMBER memb-list | ALL }

TR.53 Concrete Design Specifications

This section describes the specifications for concrete design for beams, columns and individual plate elements. The concrete design procedure implemented in STAAD consists of the following steps:

TR.53.1 Design Initiation

This command is used to initiate concrete design for beams, columns and individual plate elements. This command initiates the concrete design specification. Without this command, none of the following concrete design commands will be recognized.

Note: This command must be present before any concrete design command is used.

General Format

START CONCRETE DESIGN

Related Links

- D. To specify concrete design code and parameters (on page 1002)

TR.53.2 Concrete Design-Parameter Specification

This set of commands may be used to specify parameters to control concrete design for beams, columns and individual plate elements.

General Format

CODE design-code

parameter-name f1 { MEMBER member-list | ALL }

Where:

- design-code = the concrete design code name, which is described in the concrete chapters of the D. Design Codes (on page 1366).
- parameter-name f1 = is the concrete design parameter and corresponding value. Wherever applicable, this value is input in the current units. The UNIT command is also accepted during any phase of concrete design.
- parameter-name = refers to the concrete parameters described in D1.F.3 Design Parameters (on page 1481) for the ACI code and in various corresponding concrete chapters of the D. Design Codes (on page 1366) for all other codes.

Note: All values must be provided in the current unit system.

Related Links
TR.53.3 Concrete Design Command

This command may be used to specify the type of design required. Members may be designed as BEAM, COLUMN or ELEMENT.

Members to be designed must be specified as BEAM, COLUMN, or ELEMENT. Members, once designed as a beam, cannot be redesigned as a column again, or vice versa.

General Format

```
DESIGN { BEAM | COLUMN | ELEMENT } { memb-list | (ALL) }
```

Notes

a. Only plate elements may be designed as ELEMENT.
b. Enter this command after the parameters needed for this command have been entered.
c. The DESIGN ELEMENT command designs individual plate elements using the procedure explained in D1.F.6 Slab Design (on page 1522) for the ACI code. For theoretical information on designing individual plate elements per other design codes, please refer to D. Design Codes (on page 1366).

Tip: The D. Interactive Concrete Design (on page 1005) is used for slab design.

Related Links

• D. To specify concrete beam design command (on page 1003)

TR.53.4 Concrete Take Off Command

This command may be used to obtain an estimate of the total volume of the concrete, reinforcement bars used and their respective weights.

This command can be issued to print the total volume of concrete and the bar numbers and their respective weight for the members designed.

Tip: This command may be used effectively for quick quantity estimates.

General Format

```
CONCRETE TAKE OFF
```

Sample Output

```
************** CONCRETE TAKE OFF **************

NOTE: CONCRETE QUANTITY REPRESENTS VOLUME OF CONCRETE IN BEAMS, COLUMNS, AND PLATES DESIGNED ABOVE.

REINFORCING STEEL QUANTITY REPRESENTS REINFORCING STEEL IN BEAMS AND COLUMNS DESIGNED ABOVE.

REINFORCING STEEL IN PLATES IS NOT INCLUDED IN THE REPORTED
```

Note: The D. Interactive Concrete Design (on page 1005) is used for slab design.
QUANTITY.

TOTAL VOLUME OF CONCRETE = 4.4 CU. YARD

<table>
<thead>
<tr>
<th>BAR SIZE NUMBER</th>
<th>WEIGHT (in lbs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>261</td>
</tr>
<tr>
<td>5</td>
<td>87</td>
</tr>
<tr>
<td>6</td>
<td>99</td>
</tr>
<tr>
<td>8</td>
<td>161</td>
</tr>
<tr>
<td>9</td>
<td>272</td>
</tr>
</tbody>
</table>

*** TOTAL= 880

Related Links

• D. To generate concrete take off (on page 1004)

TR.53.5 Concrete Design Terminator

This command must be used to terminate the concrete design.
This command terminates the concrete design, after which normal STAAD commands resume.

Note: Without this command, further STAAD commands will not be recognized.

General Format

END CONCRETE DESIGN

Example

START CONCRETE DESIGN
CODE ACI
FYMAIN 40.0 ALL
FC 3.0 ALL
DESIGN BEAM 1 TO 4 7
DESIGN COLUMN 9 12 TO 16
DESIGN ELEMENT 20 TO 30
END

Related Links

• D. To specify concrete design code and parameters (on page 1002)

TR.54 Footing Design Specifications

This feature has been removed from batch mode design. Please contact the Technical Support department for further information.

Footing design may be performed using STAAD Foundation Advanced through the Foundation Design workflow (on page 1359) in the graphical interface.
TR.55 Shear Wall Design

Note: Shearwall design has been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow it but its use is not recommended.

Related Links
- G.5.3 Surface Elements (Deprecated) (on page 2322)

TR.56 End Run Specification

This command must be used to terminate the STAAD run.

This command should be provided as the last input command. This terminates a STAAD run.

General Format

```
FINISH
```

Index of Commands

The following is an alphabetic list of all STAAD.Pro commands in the STAAD Command Reference.

- A B (on page 2862)
- C (on page 2862)
- D (on page 2863)
- E (on page 2864)
- F (on page 2864)
- G (on page 2864)
- H (on page 2864)
- I (on page 2864)
- J K (on page 2865)
- L (on page 2865)
- M (on page 2865)
- N (on page 2865)
- O (on page 2865)
- P Q (on page 2866)
- R (on page 2867)
- S (on page 2867)
- T (on page 2868)
- U V W Y X Z (on page 2868)

A, B

- ALUMINUM MEMBER TAKE OFF (on page 2856)
- AREA LOAD (on page 2665)

C

- CALCULATE RAYLEIGH (on page 2790)
- CHANGE (on page 2835)
- CHECK CODE (on page 2852)
- CHECK SOFT STORY (on page 2529)
- CHECK STORY DRIFT (on page 2526)
- CONCRETE TAKE OFF (on page 2860)
- CONSTANTS (on page 2503)
- CUT OFF (on page 2539)
DEFINE 1893 (on page 2576)
DEFINE AIJ (on page 2605)
DEFINE CFE (on page 2607)
DEFINE COLOMBIAN
  1998 (on page 2571)
  2010 (on page 2573)
DEFINE DAMPING (on page 2510)
DEFINE DIRECT (on page 2643)
DEFINE ENVELOPE (on page 2837)
DEFINE IBC
  2000 or 2003 (on page 2588)
  2006 or 2009 (on page 2592)
DEFINE IMPERFECTION (on page 2511)
DEFINE MATERIAL (on page 2501)
DEFINE MEMBER ATTRIBUTE (on page 2535)
DEFINE MESH (on page 2435)
DEFINE MOVING LOAD (on page 2541)
DEFINE NRC
  1995 (on page 2548)
  2005 (on page 2552)
DEFINE NTC (on page 2610)
DEFINE PMEMBER (on page 2443)
DEFINE REFERENCE LOAD (on page 2642)
DEFINE RPA (on page 2546)
DEFINE SNOW LOAD (on page 2641)
DEFINE TIME HISTORY (on page 2630)
DEFINE UBC
  1997 (on page 2620)
  1994 or 1985 (on page 2617)
DEFINE WIND (on page 2623)
DELETE (on page 2445)
DESIGN (on page 2860)
DRAW (discontinued)

E

ELEMENT INCIDENCES (on page 2431)
ELEMENT INCIDENCES SOLID (on page 2432)

ELEMENT LOAD

JOINTS (on page 2661)
PLATE (on page 2657)
SOLID (on page 2660)

ELEMENT PLANE STRESS (on page 2498)
ELEMENT PROPERTY (on page 2487)
ELEMENT RELEASES (on page 2490)
END CONCRETE DESIGN (on page 2861)
END STEADY (on page 2817)

F

FIXED END LOAD (on page 2683)
FLOOR DIAPHRAGM (on page 2526)
FLOOR LOAD (on page 2672)
FREQUENCY (on page 2809)

G

GENERATE ELEMENT (on page 2435)
GENERATE FLOOR SPECTRUM (on page 2833)
GROUND MOTION (on page 2767)
GROUP (on page 2855)

H

HARMONIC FORCE (on page 2813)
HARMONIC GROUND (on page 2812)

I

IGNORE LIST (on page 2423)
IGNORE STIFFNESS (on page 2491)
INACTIVE (on page 2445)
Technical Reference of STAAD Commands
Index of Commands

INPUT NODesign (on page 2424)
INPUT WIDTH (on page 2413)

J, K

JOINT COORDINATES (on page 2425)
JOINT LOAD (on page 2651)

L

LOAD COMBINATION (on page 2791)
LOAD GENERATION (on page 2771)
LOAD LIST (on page 2836)
LOAD LIST ENVELOPE (on page 2837)

M

MATERIAL (on page 2503)
MEMBER CABLE (on page 2493)
MEMBER COMPRESSION (on page 2495)
MEMBER CRACKED (on page 2485)
MEMBER CURVED (on page 2475)
MEMBER FIREPROOFING (on page 2482)
MEMBER INCIDENCES (on page 2428)
MEMBER LOAD (on page 2653)
MEMBER OFFSETS (on page 2499)
MEMBER PRESTRESS LOAD (on page 2678)
MEMBER PROPERTIES (on page 2459)
MEMBER RELEASES (on page 2488)
MEMBER TENSION (on page 2495)
MEMBER TRUSS (on page 2492)
MODAL CALCULATION (on page 2791)
MODE SELECT (on page 2540)
MULTILINEAR SPRINGS (on page 2520)

N

NOTIONAL LOAD (on page 2785)
Technical Reference of STAAD Commands

Index of Commands

R

REFERENCE LOAD (on page 2789)
REPEAT LOAD (on page 2770)

S

SECTION (on page 2839)
SELECT (MEMBER) (on page 2853)
SELECT OPTIMIZED (on page 2854)
SELFWEIGHT (on page 2685)
SEPARATOR (on page 2422)
SET (on page 2413)
SIZE (on page 2849)
SLAVE (on page 2525)
SNOW LOAD (on page 2785)
SPECTRUM (on page 2688)

1893 (on page 2721) (Indian)
EURO (on page 2710), EURO 2004 (on page 2715)
IBC 2006 (on page 2743)

SPRING COMPRESSION (on page 2522)
SPRING DAMPING (on page 2511)
SPRING TENSION (on page 2522)

STAAD (on page 2409)
START DECK DEFINITION (on page 2473)
START GROUP DEFINITION (on page 2440)
START JOB INFORMATION (on page 2424)
START USER TABLE (on page 2446)
STEADY FORCE FREQUENCY (on page 2810)
STEADY GROUND FREQUENCY (on page 2809)
STEEL MEMBER TAKE OFF (on page 2856)
SUBSTITUTE (on page 2439)
SUPPORTS (on page 2513)
SURFACE CONSTANTS (on page 2508)
SUPPORT DISPLACEMENT (on page 2683)
SURFACE INCIDENCES (on page 2433)
Technical Reference of STAAD Commands

Index of Commands

SURFACE LOAD (on page 2663)
SURFACE PROPERTY (on page 2488)

T

TEMPERATURE LOAD (on page 2682)
TIME LOAD (on page 2630)

U, V, W, X, Y, Z

UNIT (on page 2411)
This section describes the tools found in the ribbon toolbars.

**Note:** The ribbon toolbar dynamically updates for the current workflow selection.

**File tab**

The File tab opens the STAAD.Pro Backstage view, which is used for file operations.

**Info tab**

Used to specify general information about the structure, including a job description, job number, persons responsible for creating, checking, and approving the structure, etc. This page also provides general information about the structure such as the total numbers of nodes, members, elements as well as the STAAD commands present in the structure file.

**Job Information**

- **Job Name**: Type a short description of the job. This description appears in the reports.
- **Client**: Type the client name, if any.
- **Job Number / Revision / Part / Reference**: Type the job, part and reference numbers, if any.
- **Engineer / Checker / Approved**: These items are used to track modifications in the structure file. The names and dates of the changes may be entered.
- **Last Review**: Type the date of the last review provided by respective engineer, checker, or approver.

**CONNECT Information**

When your STAAD.Pro model has a CONNECTED Project associated with it, the CONNECTED Project is displayed here. Click [...] to open the CONNECTED Project Information dialog which contains CONNECTED Project details and can be used to change the associated CONNECTED Project.

**File Information**

This group contains several items for information only. These items include the Filename, Directory, Date / Time of first creation, and the File size.
Model Information
Displays the total numbers of nodes, beams, and plates present in the structure. The Highest # column shows the highest Node / Beam / Plate numbers in the structure. In case the structure has discontinuous Node/Member/Element number, the Highest # value will be different from the Count value.

The Commands in File group shows the STAAD commands which have been used in the structure.

New tab
Used to create a new STAAD project. Options are available to start with an analytical model, a physical model, or in STAAD Building Planner.

The Additional License options are available here to allow you to select which license options you want to use before creating a file.

Open tab
Used to open an existing STAAD input file or Archive. You can also open STAAD projects from a ProjectWise data source.

Select the Recent tab to view recent files by type. Hover your mouse pointer over any file to display the file and CONNECT Project properties.

The Additional License options are available here to allow you to select which license options you want to use before opening a file.

Save / Save As tabs
Used to save changes to the current model or to save the current model as a different file.

Save
Click this entry to save any changes made to the current STAAD input file.

Save As
This tab allows you to enter a new File Name and Location to save the current input file as a different file. Click Browse to open a Windows open dialog to select a folder.

Click Save to save the new file.

Print tab
Used to print input, output, and reports for the current STAAD project.

Tip: Press <Ctrl+P> to open this File menu tab.
### Ribbon Control Reference

#### File tab

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Input Command File</strong></td>
<td>Used to print the contents of the STAAD input file (file extension .std) from the STAAD Editor.</td>
</tr>
<tr>
<td><strong>Output File</strong></td>
<td>Used to print the contents of the STAAD output file (file extension .anl) from the STAAD Output Viewer.</td>
</tr>
<tr>
<td><strong>Report</strong></td>
<td>Used to print the report for the STAAD input file.</td>
</tr>
<tr>
<td></td>
<td><strong>Note</strong>: You can set up the report contents in the Postprocessing workflow.</td>
</tr>
<tr>
<td><strong>Error Report</strong></td>
<td>Print error file, if any.</td>
</tr>
</tbody>
</table>

#### Report tab

Used to configure, review, and export STAAD reports.

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Setup</strong></td>
<td>Opens the Report Setup dialog, which is used to specify the contents of the customized report.</td>
</tr>
<tr>
<td><strong>Preview Report</strong></td>
<td>Opens the Report Print Preview window, which is used to review your project report before creating a hard copy.</td>
</tr>
<tr>
<td></td>
<td><strong>Note</strong>: You can also preview the report in the Report page in the Postprocessing workflow.</td>
</tr>
<tr>
<td><strong>Export to Text File</strong></td>
<td>Opens a Save As dialog, which is used to save the current report to a plain text file (.txt file extension).</td>
</tr>
<tr>
<td><strong>Export to MS Word File</strong></td>
<td>Opens a Save As dialog, which is used to save the current report to a Microsoft Office Word document (.doc file extension).</td>
</tr>
</tbody>
</table>

#### Report Setup dialog

Used to specify the contents of the customized report

Opens when the Reports page is selected in the Postprocessing workflow.

#### Items tab

The Items tab is used for specifying the items to be included in the report.

- **Report Item Categories**
  - Select a report item category from the drop-down list. Items in this category are displayed in the Available list below.

- **Available list**
  - Displays items that may be included in the report. The Available list is divided into six categories: Input, Output, Pictures, Reports, STAAD etc Output, Steel Design Output and
Advanced Query Reports. Input lists all input related data that may be included in a report. Output lists all output related data that may be included in a report. Pictures lists all the pictures included in the Picture Album. Reports lists all reports that have been previously created and saved. Thus, reports may contain sub-reports. STAAD.etc Output - if a component design such as Base Plate design, Footing design, or a Moment Connection design, using the interface with the STAAD.etc program is carried out - see Section 2.3.8 of this manual - the results of that design can be included in the final report by selecting this item. Steel Design Output - if members are designed through the Interactive Design | Steel Design, then the design output for the members are listed in this category and can be included in the final report. Advanced Query Reports are customized reports created using the facility Tools | SQL Query | Advanced Query.

**Selected list**

Displays items that have been selected for the report. The Report Detail Increments edit box is used to specify the number of segments into which a member would be broken up for printing section forces. Buttons to transfer selected items from the Available to the Selected list (> and >> ) and vice versa (< and << ) are included. Also, note that there are up and down arrows to the right of the Selected list. These buttons are used for arranging the order of the items in the report.

There are a few methods for moving items from the Available list to the Selected list. Select the type of items you wish to choose from, Input, Output, Pictures or Reports. Note that these items may be mixed and matched. They are categorized to make locating items easier. Clicking on the right-pointing double arrow button (>>) moves all items in the Available list to the Selected list. To move only selected items, select the items. To select multiple items, hold down the Control key while selecting items. Transfer the selected items to the Selected list by clicking on the right-pointing single arrow button (>). Double clicking on an item in the Available list will move it to the Selected list.

When this dialog box opens, a default item, Job Information appears in the Selected list. You may remove one or more items from the Selected list by clicking on the < or << arrow buttons.

Items in the Selected list can be rearranged in any desired order. To rearrange items, select the item to be moved up or down in the list and then use the up or down arrow to the right of the Selected list.

**Report Details**

**Increments**

**Load Cases tab**

Used specify which load case for which results are to be included in the report.

The results can be grouped by node and beam numbers or by load case. Specify the method of Grouping using the radio buttons. Load cases are moved to and from the Available and Selected lists as described before under the Items tab.

**Available list**

Displays items that may be included in the report. The Available list is divided into six categories: Input, Output, Pictures, Reports, STAAD.etc Output, Steel Design Output and Advanced Query Reports. Input lists all input related data that may be included in a report. Output lists all output related data that may be included in a report. Pictures lists all the pictures included in the Picture Album. Reports lists all reports that have been previously created and saved. Thus, reports may contain sub-reports. STAAD.etc Output - if a component
design such as Base Plate design, Footing design, or a Moment Connection design, using the interface with the STAAD.etc program is carried out - see Section 2.3.8 of this manual - the results of that design can be included in the final report by selecting this item. Steel Design Output - if members are designed through the Interactive Design | Steel Design, then the design output for the members are listed in this category and can be included in the final report. Advanced Query Reports are customized reports created using the facility Tools | SQL Query | Advanced Query.

**Selected list**
Displays items that have been selected for the report. The Report Detail Increments edit box is used to specify the number of segments into which a member would be broken up for printing section forces. Buttons to transfer selected items from the Available to the Selected list (>) and vice versa (< and <<) are included. Also, note that there are up and down arrows to the right of the Selected list. These buttons are used for arranging the order of the items in the report.

There are a few methods for moving items from the Available list to the Selected list. Select the type of items you wish to choose from, Input, Output, Pictures or Reports. Note that these items may be mixed and matched. They are categorized to make locating items easier. Clicking on the right-pointing double arrow button (>>) moves all items in the Available list to the Selected list. To move only selected items, select the items. To select multiple items, hold down the Control key while selecting items. Transfer the selected items to the Selected list by clicking on the right-pointing single arrow button (>). Double clicking on an item in the Available list will move it to the Selected list.

When this dialog box opens, a default item, Job Information appears in the Selected list. You may remove one or more items from the Selected list by clicking on the < or << arrow buttons. Items in the Selected list can be rearranged in any desired order. To rearrange items, select the item to be moved up or down in the list and then use the up or down arrow to the right of the Selected list.

**Grouping for Load Tables**

**Grouping for Result Tables**

**Modes tab**
Used to select mode shape numbers for which the results will be included for dynamic analyses (modal calculation, response spectrum, time history, etc.).

**Note:** A successful dynamic analysis must first be performed for mode shapes to be made available.

**Grouping**
Select to group modal results by Node number or by Mode number.

**Available list**
Displays available modes in the mode.

**Selected list**
Displays all modes in the model. All mode shapes will be added to the Selected list by default.

**Ranges tab**
Used to select the members and elements to be included in the report. By default, the report includes all members, elements, and nodes.
Ribbon Control Reference

File tab

**All**
Select this option to include all model entities in the report (default).

**View**
Select this option to select a previously saved view in the drop-down list. Only model entities included in the selected view will be included in the report.

**Group**
Select this option to select a previously defined group name in the drop-down list. Only model entities included in the selected group will be included in the report.

**Property**
Select this option to select a property tag in the drop-down list. Only model entities included in the selected property will be included in the report.

**Ranges**
Select this option to specify a range of **Nodes** (by node number) and/or **Beams/Plates/Solids** (by entity number). Separate individual entries in either list using a comma. Ranges of values can be represented using a dash (e.g., 5-10).

**Steel Design tab**
Used to include steel design results that have been computed during analysis.

**Check Results**
Radio buttons allow the selection of results generated by the code check facility. Use the radio buttons to specify whether to include no results, all results, the results of the first check or the results of the final check.

**Select Results**
Radio buttons allow the selection of results generated by the member selection facility. Use the radio buttons to specify whether to include no results, all results, the results of the first select or the results of the final check.

**Summary Only (Track 0)**
Set this option to include a brief design summary for a BS 5950 steel design. Otherwise, the specified level of details (TRACK command option) is used.

**Print Failed checks Only**
Set this option to include only steel design results for members which have failed one or more checks. Otherwise, the specified level of details (TRACK command option) is used for both passing and failing members.

**Font**
Opens an Font dialog, which is used to change the typeface parameters used for steel design results. The current font settings are displayed adjacent to this button.

**Picture Album tab**
Used to control the display of pictures of the structure that have been taken using the **Take Picture** tool.

**Note:** To include pictures in the report, they must be added in the **Selected** list on the **Items** tab. Pictures are available in the Pictures category.

**Name**
A list of all available pictures appears in the drop down list.

**Delete Picture**
Removes the currently selected picture from the picture album.

**Caution:** This action cannot be undone.
radio buttons allow the selection of results generated by the code check facility. Use the radio buttons to specify whether to include no results, all results, the results of the first check or the results of the final check.

Caption
Type or edit the caption to be included with the drawing in the report.

Tip: If the model has changed since the picture was taken, a note is appended to the caption letting you know that changes have been made.

Full Page
Set this option to specify values of Height and Width equal to the full size of the default paper size. The picture will then take up an entire page of the report.

Height / Width
Specify printed dimensions for the selected picture. The aspect ration of the picture will remain constant, so updating one dimension will update the other.

Click in/mm beside either dimension to toggle between the two units of length.

Width
Opens an Font dialog, which is used to change the typeface parameters used for steel design results. The current font settings are displayed adjacent to this button.

Options tab
Used for setting various report display options.

Header
Set this option to include the header on each page of the report. This includes custom name and logo, sheet numbering, job information, file name, and file modification information.

Page Outline
Set this option to add a solid, black border around the extents of the page content.

Footer
Set this option to include a footer on each page of the report. The footer contains a print time stamp, program version information, and print run information.

Prefix / Suffix
Specify optional sheet number prefix or suffix content in the respective field. This is useful if the STAAD.Pro report is to be included in an appendix or some other format where a special numbering format is used.

No. pages from
Specify the initial page number used for the report. This is useful if you have other content to include in a larger report before the STAAD.Pro output.

Reverse page order
Set this option to reverse the order in which pages are sent to the printer. The contents of the report, including sheet number, are not affected by this option.

Grid
Set this option to include grid lines around and between table elements.

Start each table on a new page
Set this option to add a page break after each table, even when a table does not fill an entire page alone.

Font
Opens a Font dialog, which is used to select the type face properties for the Column Heading or Table contents. The current font face and size are displayed for each setting.

Name and Logo tab
Used to add the your company name and/or logo to the report. This tab contains a viewing area to preview how the company name and logo will appear on the report.
**Note:** The **Remove Bentley (B) logo from Report** must be set on the **Configure** dialog File Options tab to remove the Bentley logo from the report heading.

**Preview area**  
To use a text header, simply type in the preview area. Use the **Font** and **Alignment** controls to edit the appearance of the text.
To use a graphic logo, use the **File** and **Position** controls.

**File**  
Opens a Windows file open dialog, which is used to select a Windows Bitmap image (file extension .bmp) to use graphic for inclusion in report headings.

**Paste**  
Adds an image file from the Windows clipboard.

**Delete**  
Removes the graphic file from the report header.

**Position**  
Select the horizontal position in the company identification portion of the report header.

**Font**  
Opens a Font dialog, which is used to select the type face properties for the logo.

**Tip:** In order to prevent clipping of the text block, a 14 point font size is the largest size for two lines and a 9 point font size is the largest size for three lines.

**Load / Save tab**

Used to save the contents of the customized report and to load previously saved reports.

**Report**  
Previously saved reports are available in the drop-down list. Select one to load the report details for editing, printing, or deletion.

**Save As**  
Opens the **Save Report** dialog, which is used to provide a name for the current report settings. Click **OK** in this dialog to save the named report in the **Report** list.

**Delete**  
Removes the selected report name in the **Report** list.

**Caution:** This action cannot be undone.

**OK**  
Saves any changes to the Report setup and closes the dialog.

**Cancel**  
Closes the dialog without saving any changes.

**Print**  
Prints the report with the current settings.

**Help**  
Opens the STAAD.Pro Help window.

**ISM tab**

Contains tools for working with Integrated Structural Modeling repositories.

**Note:** ISM has some limited functionality when working with analytical models. It is recommended to use the Physical Modeling workflow in order to exchange physical model data with an ISM repository through the STAAD.Pro Physical Modeler.
### Create Repository
Transfers the current model opened in STAAD.Pro and generates a new ISM Repository. This is the most common way in which an ISM Repository is initially created.

### New from Repository
Creates a new STAAD.Pro input file from an existing ISM Repository. This is used to transfer model data into other tools used for your workflow.

### Update Repository
Coordinates changes made to the model in the client application and coordinate some or all of those changes with an existing ISM Repository.

### Update from Repository
Updates the current STAAD.Pro input file with some or all of the changes which have been made to the ISM Repository.

---

## Import/Export tab

### Import
Used to import data from other formats. Opens a Windows Open dialog, which is used to select a file name and location for importing the selected file format.

- **DXF (*.dxf)** Used to import an Autodesk AutoCAD® 3D DXF file.
- **QSE ASA (*.asa)** Import an ASA format file.
- **Stardyne (*.sdn)**
- **CIS/2 (*.stp)** Options are provided for creating new models (from Start page only) or updating the current model (when a STAAD project is open) from a CS/2 model.

### Export
Used to export data to other formats, such as DXF, VRML, the CIMsteel STEP format, etc.

- **3D DXF (*.dxf)** Exports the entire model into a 3D DXF file.
- **2D DXF (*.dxf)** Exports a selected plane of the model into a 2D DXF file. This option opens the Export 2D DXF dialog.

  **Note:** The Export 2D DXF dialog is used to indicate the section of the model to use to generate a plan as well as the model objects to include in the drawing.

- **QSE ASA (*.asa)**
- **VRML (*.vml)** Exports the model as a 3D vector graphic in the Virtual Reality Modeling Language World file. This option opens the VRML2 Options dialog, which is used to set the colors for the World file.

  **Note:** The VRML2 Options dialog is used to specify the colors in the exported file.

- **CIS/2 (*.stp)**
DXF Import dialog

Used to specify the vertical axis for data imported from a DXF file into a STAAD input file.

Opens when a DXF file is selected from the Import dialog.

Structure Convention

Select an option for defining the global vertical ("up") axis for the structure.

- **No Change** - Use the vertical axis as defined in the 3D DXF file.
- **Y Up** - Adds a `SET Y UP` command to the STAAD input file.
- **Z Up** - Adds a `SET Z UP` command to the STAAD input file.

**OK**

Imports the selected DXF file contents into the current STAAD input file.

**Cancel**

Closes the dialog without importing any model information.

**Help**

Opens the STAAD.Pro help window.

Cloud Services tab

Contains tools for using Bentley Cloud Services.

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Associate Project</td>
<td>Used to associate your model with a ProjectWise Project.</td>
</tr>
<tr>
<td>Disassociate Project</td>
<td>Used to disassociate your model with the current ProjectWise Project. If you wish to change the associated ProjectWise Project, you do not need to first disassociate the model first.</td>
</tr>
<tr>
<td>CONNECTION Center</td>
<td>Opens the CONNECT Personal Portal in your web browser.</td>
</tr>
<tr>
<td>ProjectWise Projects</td>
<td>Opens the Project Portal for your ProjectWise Project.</td>
</tr>
<tr>
<td>Save Project Settings</td>
<td>Saves all STAAD.Pro configuration files as ProjectWise Project settings.</td>
</tr>
<tr>
<td>Scenario Services</td>
<td>Opens the Scenario Services portal in your web browser.</td>
</tr>
</tbody>
</table>

Settings tab

Used to control some of the application settings.
### Display Options

Opens the **Options** dialog, which is used to change the length and force units for various values which can be displayed in the graphics window, such as member properties, material constants, load magnitudes, plate stresses, etc.

### Set Structure Colors

Opens the **Color Manager** dialog, which is used to set colors for highlight, entities, and results.

### Structural Tooltip Options

Opens the **Tool Tip Options** dialog, which is used to set tooltip options for structural elements.

---

**Tools tab**

Contains tools for managing file backups.

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open Backup Manager</td>
<td>Opens the <strong>Backup Manager</strong> dialog, which is used to automatically save the current STAAD file at periodic intervals.</td>
</tr>
</tbody>
</table>

**Backup Manager** dialog

Use this option to automatically save the current STAAD file at periodic intervals. The Backup Manager is also capable of restoring older versions of the current STAAD file as well as find the differences between two versions of the same or different STAAD files. When this option is selected, the dialog box shown in the next figure appears.

Opens when **Open Backup Manager** is selected from the **Tools** tab on the Backstage view.

**Enable Auto Save**  In order to activate the Auto Save facility, check this box.

**No of Auto-Save Backup**  The number of automatic saves that will transpire within the allotted Frequency. For example, if this number was 3 and the Frequency was set to 10 Min, the program would automatically save the current model every 10 minutes and on the 4th save (at 40 minutes), it would overwrite the first save (which occurred at the 10th minute).

**Frequency of Auto-Save**  How often the Auto-Save should take place (in minutes).
Ribbon Control Reference

File tab

No of Manual-Save Backup
The manual save occurs every time the user clicks on the Save button. This is independent of the Auto-Save. The distinction between the two saves is listed under the Backup-Type column. The number of manual saves that will actually be archived can be provided here. Although pressing Save four times will update your current model, if the number of Manual-Saves is set to four, there will be four revisioned versions of the STAAD file that can be restored. On the fifth save (in this example), the first archive is overridden.

Use Project Directory to keep Backup File
Check this box to keep the archived or versioned STAAD files (created using the Auto-Save or manual Save) in the same directory as the current STD file. The archived files are designated with the extension .SBK, but are recorded in binary format and cannot be viewed using the STAAD Editor. Uncheck this box and use the Change button to designate another directory to keep the backup files.

Current Backup Directory
It shows us the path or the directory name, where the backup files are being saved at present.

Change
Click on this button to change the current backup directory. The user can browse and locate the folder where the backup files can be saved.

File Name
A list of all the backup files of the saved file.

Backup Type
It shows us the type of the backup file. If the file is saved automatically, then it will be shown as an "Auto" backup type; else, it will be shown as a "Manual" backup type.

Compare
Compares the difference between two saved or archived versions of a STAAD file. Select the two STAAD files under the File Name column (press <CTRL>) and click on the Compare button. The differences will be highlighted using blue and green colors within a makeshift Editor.

Save As
Select a saved version of the current STAAD file and save it as a STD file in another directory.

Open...
Will open the selected saved version of the current STAAD file in the current session. This will not overwrite the current STAAD model. It will open the saved file by restoring it to the "temp" directory.

Restore
Will restore and overwrite the current STAAD file/model with the selected saved STAAD file.

Help tab
Contains links for getting additional assistance and information about STAAD.Pro.

Table 277: Help menu items

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td>Opens the online help in your web browser.</td>
</tr>
<tr>
<td>OpenSTAAD Help</td>
<td>Opens the online help in your web browser to the OpenSTAAD reference section.</td>
</tr>
<tr>
<td>Technical Support</td>
<td>Opens the Worldwide Technical Support Resources dialog, which offers a map of worldwide technical support contacts for STAAD.Pro.</td>
</tr>
</tbody>
</table>
## Table 278: About menu items

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>About STAAD.Pro</td>
<td>Opens the About STAAD.Pro CONNECT Edition dialog, which contains version, licensing, and legal information about the product. Any Technical Preview items contained in the current version will also be listed here.</td>
</tr>
<tr>
<td>Product News</td>
<td>Opens the STAAD.Pro product page at Bentley.com in your web browser.</td>
</tr>
<tr>
<td>Home Page</td>
<td>Opens the Bentley.com home page in your web browser.</td>
</tr>
</tbody>
</table>

## Geometry tab
## Table 280: Clipboard group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
</table>
| 🔄 Undo    | Negates the last operation.  
**Note:** You cannot undo settings changes.  
After you undo an operation, the operation previous to the negated operation can be undone. You can, therefore, undo a series of previous operations by repeatedly choosing Undo.  
There is no limit to the number of undos that can be performed.  
**Note:** Only geometry, property, support and member/plate specification operations support Undo/Redo. | <Ctrl+Z> |
| ⏩ Redo    | Negates the last undo operation.  
You can redo a series of negated operations by repeatedly choosing Redo.  
Each undo operation is a single redoable operation regardless of the number of negated operations. | <Ctrl+Y> |
| ✗ Delete   | Used to delete selected object(s). | <Del(ete)> |
| 🪓 Cut      | Used to cut selected object(s) (delete and copy to clipboard). The deleted objects may then be pasted.  
The object(s) remain on the Clipboard until another Cut or Copy is performed. | <Ctrl+X> |
## Ribbon Control Reference

### Geometry tab

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Copy</strong></td>
<td>Used to copy selected object(s) to clipboard for subsequent pasting. The element(s) remain on the Clipboard until another Copy or Cut is performed.</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
<tr>
<td><strong>Paste</strong></td>
<td>Opens the Paste with Move dialog (on page 2891), which is used to specify the insertion point for pasting clipboard elements into the model. Because the elements remain on the Clipboard after pasting, you can paste repeatedly.</td>
<td>&lt;Ctrl+V&gt;</td>
</tr>
</tbody>
</table>

Table 281: Structure group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Grids &gt;</strong></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Beam Grid" /></td>
<td>Opens the Snap Node/Beam dialog, which is used to specify the grid and snap settings as well as to create members and nodes automatically by snapping to grid points.</td>
</tr>
<tr>
<td><img src="image" alt="Triangular Plate Grid" /></td>
<td>Opens the Snap Node/Plate dialog, which is used to specify the grid and snap settings as well as to create plates and nodes automatically by snapping to grid points.</td>
</tr>
<tr>
<td><img src="image" alt="Quad Plate Grid" /></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Solid Grid" /></td>
<td>Opens the Snap Node/Solid dialog, which is used to specify the grid and snap settings as well as to create solids and notes automatically by snapping to grid points.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Structure Wizard</td>
<td>Opens the <strong>Structure Wizard</strong> window (on page 710), which offers a library of pre-defined structure prototypes, such as Pratt Truss, Northlight Truss, Cylindrical Frame, etc. The <strong>Structure Wizard</strong> may parametrically generate a structural model, and then transfer and superimpose it on the current structure.</td>
</tr>
<tr>
<td>Translational Repeat</td>
<td>Opens the <strong>Translational Repeat</strong> dialog, which is used to make copies of portions of the structure at intervals along a linear path.</td>
</tr>
<tr>
<td>Circular Repeat</td>
<td>Opens the <strong>3D Circular</strong> dialog, which is used to make copies of portions of the structure at intervals along a circular path.</td>
</tr>
<tr>
<td>Rotate</td>
<td>Opens the <strong>Rotate</strong> dialog, which is used to rotate the selected portions of the structure about the specified axis through a specified angle. The selected portions may be copied or moved.</td>
</tr>
<tr>
<td>Mirror</td>
<td>Opens the <strong>Mirror</strong> dialog, which is used to copy or move the selected portions of the structure by “mirroring” about any plane parallel to one of the three global Cartesian coordinates.</td>
</tr>
<tr>
<td>Wall/Slab Connection</td>
<td>Used to select a wall which will have a common boundary with a slab consisting of a previously generated finite element mesh. This tool allows the consideration of boundary conditions at the interface of the panel and any other panel on whose surface one of its edge lies.</td>
</tr>
<tr>
<td>Input Units</td>
<td>Opens a drop-down list used to set the current input units for length and force. Click <strong>Apply</strong> to set the unit selection.</td>
</tr>
</tbody>
</table>
Table 282: Node group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Move Node &gt;</td>
<td><strong>Move Joint</strong> Opens the Move Entities dialog, which is used to specify the translational offset for moving a selection of nodes.</td>
</tr>
<tr>
<td></td>
<td><strong>Move Origin</strong> Opens the Move Origin dialog, which is used to specify the translational offset for moving a the model origin. All model</td>
</tr>
<tr>
<td></td>
<td>geometry is updated relative to the new origin location.</td>
</tr>
<tr>
<td>Merge Selected Nodes</td>
<td>Opens the Select Node dialog, which is used to select a node number you want to keep in the model. Other selected nodes will be merged to</td>
</tr>
<tr>
<td></td>
<td>this node number.</td>
</tr>
<tr>
<td>Renumber Nodes</td>
<td>Opens the Renumber dialog, which is used to renumber selected nodes starting with a specified number. The numbering sequence can be in</td>
</tr>
<tr>
<td></td>
<td>an ascending or descending order and the order can be sorted by some criteria if needed.</td>
</tr>
</tbody>
</table>

Table 283: Beam group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add Beam &gt;</td>
<td><strong>Add Beam</strong> Used to add beam by clicking start and end nodes.</td>
</tr>
<tr>
<td></td>
<td><strong>Add Beam Between Midpoints</strong> Used to add a new beam connecting the midpoints of two existing beams. New nodes will be created at the</td>
</tr>
<tr>
<td></td>
<td>exiting beam midpoints.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
</tr>
<tr>
<td>Add Beam by Perpendicular Intersection</td>
<td>Used to add a beam from an existing node perpendicular to an existing beam. A new node will be created on the existing beam.</td>
</tr>
<tr>
<td>Add Curve Beam</td>
<td>Used to define curved members. The curve must be a segment of a circle and the internal angle subtended by the arc must be less than 180 degrees.</td>
</tr>
<tr>
<td>Create Colinear Beams</td>
<td>Used to create beams between selected nodes which lie along a straight path (i.e., colinear).</td>
</tr>
<tr>
<td>Set New Beam Attributes</td>
<td>Opens the <strong>Define Member Attributes</strong> dialog, which is used to predefine member attributes such as properties, materials, beta angles, or release specifications for member attribute sets. Members which are then added to the model will have these properties by default.</td>
</tr>
<tr>
<td>Beam Layout</td>
<td>Opens the <strong>Beams</strong> table.</td>
</tr>
<tr>
<td>Insert Node</td>
<td>Opens the <strong>Insert Node into Beam #</strong> dialog, which is used to insert one or more nodes at specified distances along selected members.</td>
</tr>
<tr>
<td>Stretch Members</td>
<td>Opens the <strong>Stretch Member(s)</strong> dialog, which is used to increase the length of a member in various ways.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------------------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td> Connect Beams</td>
<td>**Connect Beams Along X</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Move Beam</td>
<td>Opens the Move Selected Beams dialog, which is used to specify the translational offset for moving a selection of beams.</td>
</tr>
<tr>
<td>Intersect Beams &gt;</td>
<td><strong>Highlight Intersect Beams</strong> Used to scan the model and locate members which cross each other in space, but are not necessarily connected to each other at the intersection point.</td>
</tr>
<tr>
<td>Intersect Selected Beams</td>
<td><strong>Intersect Selected Beams</strong> Splits members at detected intersection points and creates an addition node at this intersection point such that the members are now connected.</td>
</tr>
<tr>
<td>Merge Selected Beams</td>
<td>Opens the Merge Selected Beams dialog, which is used to join two collinear beams and replace them with one beam.</td>
</tr>
<tr>
<td>Renumber Beams</td>
<td>Opens the Renumber dialog, which is used to renumber selected beams starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.</td>
</tr>
<tr>
<td>Break Beams at Selected Nodes</td>
<td>Splits a beam into separate beams at any selected node(s) which are located along that beam. This is useful when some new nodes are created on the line of the member and you want to split the beams at these newly created nodes.</td>
</tr>
</tbody>
</table>
Table 284: Plate group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add Plate &gt; Add Quad Plate</td>
<td>Used to add Quadrilateral plates. To create new elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the plate being generated.</td>
</tr>
<tr>
<td>Add Triangle Plate</td>
<td>Used to add triangular plates. To create new elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the plate being generated.</td>
</tr>
<tr>
<td>Create Infill Plates</td>
<td>Used to automatically generate plates from a selection of beams bounding panels. Panels bounded by beams on all sides are filled with plates. Typically used for floor slabs, this method significantly reduces the modeling time for generation of floor slabs in multi-storied framed structures. If no closed polygon can be found of the enclosed shape is not planar, then an error message will be displayed.</td>
</tr>
<tr>
<td>Set New Plate Attributes</td>
<td>Opens the Define Plate Object Property dialog, which is used to define the property, material and releases to each new plate element as it is created.</td>
</tr>
<tr>
<td>Plate Layout</td>
<td>Opens the Plates table.</td>
</tr>
<tr>
<td>Parametric Models</td>
<td>Opens the Parametric Models dialog, which is used to create and edit wall, slab, and panel meshes.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Generate Mesh</strong></td>
<td>Used to create a collection of plates by selecting node points and then defining mesh parameters.</td>
</tr>
<tr>
<td>Create Mesh</td>
<td>Used to generate a finite element mesh for an existing plate element.</td>
</tr>
<tr>
<td></td>
<td>The polygon can be meshed into quadrilateral or triangular elements. You have control over parameters like number of divisions along each side of the polygon. Polygonal holes can also be defined within the surface during the meshing process for polygonal meshing.</td>
</tr>
<tr>
<td><strong>Plate Mesh</strong></td>
<td>Open the <strong>Move Selected Plates</strong> dialog, which is used to specify the translational offset for moving a selection of plates.</td>
</tr>
<tr>
<td><strong>Move Plate</strong></td>
<td>Open the <strong>Renumber</strong> dialog, which is used to renumber selected plates starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.</td>
</tr>
</tbody>
</table>
## Table 285: Solid group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="102x102" alt="Add Solid &gt;" /></td>
<td>Click to add a solid of 8, 7, 6, 5, or 4 nodes, respectively.</td>
</tr>
<tr>
<td><img src="102x102" alt="Solid Layout" /></td>
<td>Opens the <strong>Solids</strong> table.</td>
</tr>
<tr>
<td><img src="102x102" alt="Move Solid" /></td>
<td>Opens the <strong>Move Selected Solids</strong> dialog, which is used to specify the</td>
</tr>
<tr>
<td></td>
<td>translational offset for moving a selection of solids.</td>
</tr>
<tr>
<td><img src="102x102" alt="Renumber Solids" /></td>
<td>Opens the <strong>Renumber</strong> dialog, which is used to renumber selected solids</td>
</tr>
<tr>
<td></td>
<td>starting with a specified number. The numbering sequence can be in an</td>
</tr>
<tr>
<td></td>
<td>ascending or descending order and the order can be sorted by some criteria</td>
</tr>
<tr>
<td></td>
<td>if needed.</td>
</tr>
</tbody>
</table>
Table 286: Physical Member group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Physical Member Layout</td>
<td>Opens the <strong>Physical Members</strong> table.</td>
</tr>
<tr>
<td>Toggle Physical Member Mode</td>
<td>Sets the analytical modeling mode to assign sections, properties, and design parameters to physical members.</td>
</tr>
<tr>
<td>Form Members</td>
<td>Used to manually form a physical structural member from a selection of one or more connected analytical beam segments.</td>
</tr>
<tr>
<td>AutoForm Members</td>
<td>Used to automatically generate multiple physical members from a group of selected analytical beams.</td>
</tr>
</tbody>
</table>

Table 287: Composite Deck group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Composite Deck Layout</td>
<td>Opens the <strong>Composite Deck</strong> dialog, which is used to define and assign composite floor deck systems.</td>
</tr>
</tbody>
</table>

**Paste with Move** dialog

Used to specify the insertion point for pasting clipboard elements into the model.

Opens when the **Paste** tool is selected with one or more objects on the clipboard.
By distance between following two nodes

Specify two existing nodes. The vector distance between these two nodes will be used to insert a copy with respect to the original.

By the following X, Y, and Z values

Specify global axis distances to insert a copy to, with respect to the original.

OK

Accepts the paste settings and inserts a copy into the model.

Cancel

Closes the Paste with Move dialog without inserting a copy.

Help

Opens the Help window.

Reference Pt

Opens the Specify Reference Point dialog, which is used to select the insertion point which the paste command will be executed.

Translational Repeat dialog

Used to make copies of portions of the structure at intervals along a linear path

Opens when the Repeat > Translational tool is selected from the Beam group on the Geometry ribbon tab.
Global Direction  Choose any one of the three possible global directions along which the selected structural entities should be copied.

No of Steps  Specify the number of copies you want.

Default Step Spacing  Type the default spacing between steps (or copies) in the edit box in current length units. You can change the spacing between copies individually using the step table.

Step Spacing table  You can manually type the Spacing to each step as well as type the Number From to use when the Renumber Bay option is selected.

Renumber Bay  Instructs the program to use a user-specified starting number for the members generated in each step of the repeat.

Link Steps  Instructs the program to generate new members between each step in the direction of the repeat.

Open Base  This option instructs the program to not generate linking members at the base of the structure (i.e., the lowest nodes in the selection).

Generation Flags  Select the structural data that you want copied to the newly generated elements:

- **All** — Uses all properties assigned to the selected object (e.g., loads, properties, design parameters, member releases, etc.) are copied
- **Geometry Only** — Only geometry data is copied. That is, nodes, member and element incidences are generated but no properties, loads, etc. will be assigned to the new objects.
- **Geometry and Property Only** — Only geometry data and property assignments are copied. All other assignments (e.g., loads and design parameters) are not copied.

Related Links
- *M. To generate copies of geometry along a line* (on page 702)

**3D Circular dialog**

Used to make copies of portions of the structure at intervals along a circular path.
Opens when the Repeat > Circular tool is selected from the Structure group on the Geometry ribbon tab.

**Axis of Rotation**  Select the global axis of rotation about which the selected components are repeated.

**Through group**

- **Node**  The reference point through which the axis for circular repeat operation passes.
  
  Click the node selection tool and then click on a node in the view window to select a node in lieu of typing the node number.

- **Point**  Type the X Coordinate and Y Coordinate of a an arbitrary reference point (i.e., when you want to use a axis of rotation that does not pass through an existing node).

- **Use this as Reference Point for Beta angle generation**  The reference point used for the circular repeat will also be used as the reference point for the beta angle for each member. This allows you to orient all the members towards the central axis of rotation.

**Total Angle**  Type the total sweep angle of rotation between the original structure and the last copied structure.

**No of Steps**  Type the number of copies you want to generate over the specified angle.

**Geometry Only**  Only geometry data is copied. That is, nodes, member and element incidences are generated but no properties, loads, etc. will be assigned to the new objects.

**Link Steps**  Instructs the program to generate new members between each step in the direction of the repeat.

**Open Base**  This option instructs the program to not generate linking members at the base of the structure (i.e., the lowest nodes in the selection).

**Related Links**

- *M. To generate copies of geometry along an arc* (on page 702)
**Rotate** dialog

Used to rotate the selected portions of the structure about the specified axis through a specified distance. The selected portions may be copied or moved.

Opens when the **Rotate** tool is selected in the **Structure** group on the **Geometry** ribbon tab.

![Rotate dialog](image)

**Note:** Unlike the **Repeat > Circular** tool, the **Rotate** tool allows you to generate copies about any arbitrary axis instead of only one parallel to the global axes.

- **Angle**
  - Type the angle of sweep.
- **Axis Passes Through**
  - Define the axis of rotation by either a pair of existing nodes or two arbitrary points.
- **Generate Mode**
  - The **Copy** option generates new structural elements. The **Move** option changes the coordinates of the selected structural elements.
- **Link Bays**
  - (Available only for **Copy** option) This option connects the copy with the original geometry by creating new linking members between them.

**Related Links**
- *M. To rotate selected entities* (on page 701)

**Snap Node-Beam** dialog

Used to provide a construction grid which can then be used to easily draw beams, plates, or solid elements. Multiple grids may be created. Additionally, this dialog can be used to import a DXF file and use it as a template for a grid or import grid files created in another STAAD.Pro model.

**Note:** In STAAD.Pro 2007 (and later), grids are saved as separate files (with a `.grd` file extension) for re-use in other models.

Opens when the **Grids > Snap Grid Beam** tool is selected in the **Structure** group on the **Geometry** ribbon tab.

- **Grid list**
  - Lists all grid definitions in the current STAAD.Pro project.
The grid which is set to active has a check before its name. Only one grid may be active at a time.

Create

Opens the **Grid Creation Method** pop-up dialog, which can be used to define either a [Linear](#) (on page 2897), [Radial](#) (on page 2897), or [Irregular](#) (on page 2898) grid pattern.

Edit

When a grid definition is selected in the list, clicking this button opens the **Grid Creation Method** pop-up dialog for editing the definition.

Delete

Delete a selected grid definition.

Copy

Opens the **Provide Modified Grid Name** dialog, which is used to create a duplicate of the selected grid. Type the new grid name and Click **OK**. Click Cancel to close the dialog without creating a grid copy.

Rename

Opens the **Provide Modified Grid** Name dialog, which is used to rename the selected grid. Type the new grid name and Click **OK**. Click Cancel to close the dialog without renaming the grid.

Import

Opens the Import Options dialog, which is used to specify the imported grid file type. DXF files or STAAD.Pro Grid files (file extension .grd) may be used.

- **DXF** — This option opens the **Import Drawing Grid** dialog (on page 2899), which is used to select elements from a .DXF file for use as a construction grid in STAAD.Pro. Placement of an imported grid in the view window is controlled using the **Import Grid** (on page 2900) pattern.
- **STAAD Grid (.grd)** — Used to Import Grids previously defined in another STAAD.Pro model. Selecting this option opens a browse dialog box to identify a .GRD file created by the Snap Node Grid tool.

Active Grid Labels Setup

- **Local Coordinate** - this option allows us to view the grid line setting in the local coordinate mode. In this case, the construction grid lines intersecting the origin will be displayed as number zero construction lines and the other construction lines numbered accordingly irrespective of what was set in the global coordinate system.

- Roaming grid labels help to identify the relative and exact gridline coordinates for the construction grid lines. This is an invaluable tool especially when the gridlines are skewed about a plane and the coordinates for the gridlines need to be displayed. These grid lines are used when adding nodes, beams, plates or solids by simply snapping on the intersection of gridlines.

- Checking the **Rel Coords** box displays how far (either positive or negative) the grid line is from the origin. A “+” or “-” label will be drawn next to the gridline. By checking the **Axis Ids** box, the global axis id (X, Y or Z) which is parallel to the grid line being labeled will be drawn. The display font (size, color and type) can be changed through the **Font** button.

- To display the actual coordinates for each grid lines in reference to the origin, choose which ends (Start, End or Both) of the grid line are to be labeled. Since the construction grid lines are only drawn in a plane, there will be a maximum of two directional gridline vectors that can be labeled (XY, XZ, or YZ). The frequency at which the grid labels are drawn can be controlled through the **Freq** spin button. If the frequency is set to 3, the grid labels will only be drawn at every third gridline. The coordinate values in the X, Y and Z directions can be displayed by pressing the appropriate coordinate labels next to the frequency spin button.

Snap to existing nodes too

Select this option to select existing nodes for creating the members. If the mouse is clicked close to an existing node, the existing node is selected instead of creating a new node at that location.
Snap Node/Beam

Click to create members by snapping to grid and/or existing nodes. The button is depressed when this function is active. Click again to stop adding beams.

Related Links
- M. To add a grid for drawing objects (on page 639)

Linear

The Linear tab is meant for placing the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board.

Related Links
- M. To add a grid for drawing objects (on page 639)

Radial grid dialog

Used to generate construction lines in transformed radial coordinates. This is used to create circular type models where members are modeled as piece-wise linear straight line segments.

**Note:** Though the parameters are entered in radial coordinate notation, the result coordinate values are in Cartesian coordinates. The grid is drawn with straight lines between points along an arc as well. Members drawn between two points on an arc will be straight line. Refer to the Curved Beam tool to add a beam with a radial curve.

Opens when Radial is selected as the type of new grid or an existing radial grid is edited in a Snap Node/<entity> dialog.

- **Name**
  (Optional) Specify a name by which you can identify the grid in the Snap Node/<entity> dialog.

- **Plane**
  Used when the grid is to be placed in a global plane. Select the global plane in which the gridlines will lie.

- **Angle of Plane**
  Choose one of the three options (X-X, Y-Y, or Z-Z) and provide the angle of rotation (in degrees) of the grid plane about that axis.

- **Origin**
  The (0,0) position on the grid in the grid coordinate system. Usually, the grid origin coincides with the structure origin.
  Type in the location of the origin of gridlines in global coordinate system, in the current length units, or click the tool to change the location of the grid origin by graphically selecting an existing node.

- **Construction Lines**
  Start Angle, is the angle in degrees about the orthogonal axis to the plane from the axis first referred to in the definition of the plane. E.g. if the selected plane is X-Y, then the angle is measured about the Z axis (using the right hand rule) from the axis parallel to the X axis.
  Sweep is the angle in degrees measured from the start angle which is divided into the selected number of Bays, thus
Figure 324: The nomenclature used for Radial grid construction lines

Related Links
- M. To add a grid for drawing objects (on page 639)

Irregular grid dialog

Used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane.

Opens when Irregular is selected as the type of new grid or an existing irregular grid is edited in a Snap Node/<entity> dialog.

Name
(Optional) Specify a name by which you can identify the grid in the Snap Node/<entity> dialog.

Origin
The (0,0) position on the grid in the grid coordinate system. Usually, the grid origin coincides with the structure origin.

Type in the location of the origin of gridlines in global coordinate system, in the current length units, or click the tool to change the location of the grid origin by graphically selecting a existing node.

Use Arbitrary Plane
This option is used when the grid is to be placed in any arbitrary plane determined by a pair of vectors, which are specified by provided a pair of coordinates entered in global X, Y, and Z values. The vectors are then taken from the origin to each of nodes to define the plane’s X and Y axis.

Tip: These coordinates do not need to be existing nodes. Any arbitrary point may be type in.

Plane
Used when the grid is to be placed in a global plane. Select the global plane in which the gridlines will lie.
Choose one of the three options (X-X, Y-Y, or Z-Z) and provide the angle of rotation (in degrees) of the grid plane about that axis.

Specify the distances between gridlines, with each distance separated by a space. Values are in the current input units of length. A gridline is always placed at zero in each direction, so there is no need to enter that value. Tabbing (or clicking) between fields updates the grids (if this is the active grid).

For example, the following X and Y distances provide a grid for drawing US.1. Plane Frame with Steel Design.std (in units of feet, with zero angle and origin = 0,0,0).

```
5 2.5 2.5 5 5 2.5 2.5 5
20 15 3 3 3
```

Related Links

- *M. To add a grid for drawing objects* (on page 639)

**DXF Import dialog**

Used to select drawing elements from a .DXF file for use as construction grids in STAAD.Pro. Opens when the .DXF option is selected in the **Import Options** dialog.
To select a DXF file click on the [...] button and navigate to the required file. The file will be opened and displayed in the preview window. Individual layers can be turned on and off from the Layers droplist. The individual entities in the selected layers are displayed and can be toggled on or off for import.

By clicking on an entity in the graphical window, the entity is highlighted in the table so that it can be turned off if required.

With the required entities selected, a suitable reference name supplied and unit selected, click on the [Import] button.

**Related Links**
- *M. To import a DXF file as a grid* (on page 641)

**Imported Grid dialog**

The data will be imported in the plane in which it was defined in the original DXF. However, if required this can be rotated about any of the global axes. Also, the origin of the grid can be located at any 3D coordinate.
The DXF grid is used just as a manually generated grid. Nodes can be created at the ends and intersections of grid lines.

**Note:** Curved lines are *not* imported.

### Angle of Plane
Choose one of the three options (X-X, Y-Y, or Z-Z) and provide the angle of rotation (in degrees) of the grid plane about that axis.

### Grid Origin
Provide the location of the origin of gridlines in global coordinate system in the current length units. One may do this by typing in the coordinates of the origin in the windows provided for the purpose.

The grid origin is the (0,0) position on the grid in the grid coordinate system. Usually, the grid origin coincides with the structure origin.

Click the tool to change the location of the grid origin by graphically selecting a existing node.

### Hide DXF text
Select this option to toggle the display of grid labels if they start clashing with the rest of the model.

**Related Links**
- *M. To import a DXF file as a grid* (on page 641)

## Mirror dialog

Used to copy or move the selected portions of the structure by “mirroring” about any plane parallel to one of the three global Cartesian coordinates.

Opens when the Mirror tool is selected in the Structure group on the Geometry ribbon tab.

### Mirror Plane
Choose one of the three global planes to mirror the selected geometry about.
**Plane Position**  
Used to establish the mirror plane position relative to the selected geometry. Select or type a **Node on Plane** to use an existing node to define the mirror plane.  
Alternately, type a distance from the global origin to the plane along the axis perpendicular to the selected Mirror Plane in the **Plane at X/Y/Z** field.

**Mirror Member Orientation**  
Select this option to consider the member orientation (Beta angle or member reference point) during mirroring. When this option is selected, the program will attempt to mirror the member orientation also, in addition to the member geometry.

**Generate Mode**  
The **Copy** option generates new structural elements. The **Move** option changes the coordinates of the selected structural elements.

**Related Links**  
- *M. To generate mirror copies of model entities* (on page 703)

### Move Entities dialog

Used to specify the translational offset for moving a selection of model objects.

Opens when the **Move Node** tool is selected in the **Node** group on the **Geometry** ribbon tab.

**Note:** The controls in this dialog are analogous to the **Move Beams Selection** dialog (opens with the **Move Beam** tool is selected), the **Move Plates Selection** dialog, and the **Move Solids Selection** dialog.

- **By distance between following nodes**
  - uses two node numbers and determines the vector between them. The selected entities are moved along a parallel vector of the same distance

- **By following X, Y and Z values**
  - uses the Cartesian coordinates specified to create a vector

- **Retain connections**
  - Set this option to retain the connectivity between objects. The connection objects will distort by the resulting translation vector.
**Note:** The model geometry will be adjusted to some degree, even if this is not selected, when nodes are the entities being moved as they member, plate, surface, and solid geometry is defined by node location. However, if you are moving a member, plate, surface, or solid, then selecting this option will move the associated nodes and thus retain the connection with adjacent entities (though their geometry will stretch, warp, or shift as a result of the move).

**Related Links**
- *M. To move selected objects* (on page 699)

**Move Origin** dialog

Used to specify the translational offset for moving a the model origin.

Opens when the **Move Origin** tool is selected in the **Node** group on the **Geometry** ribbon tab.

**Note:** All model geometry is updated relative to the new origin location.

![Move Origin dialog](image)

**Note:** The controls in this dialog are analogous to the **Move Beams Selection** dialog (opens with the **Move Beam** tool is selected), the **Move Plates Selection** dialog, and the **Move Solids Selection** dialog.

**By distance between following nodes**
Uses two node numbers and determines the vector between them. The origin is shifted along a parallel vector of the same distance.

**By following X, Y and Z values** uses the Cartesian coordinates specified to create a vector

**Related Links**
- *M. To move the model origin* (on page 700)
Select Node dialog

Used to select a node number you want to keep in the model. It opens when the Merge Nodes tool is selected in the Nodes group on the Geometry ribbon tab.

Node To Keep  The nodes in the selection set are listed. Select the node number you want to keep. Other selected nodes will be merged to this node number.

Renumber dialog

Used to renumber selected entities starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed. It opens when one of the following tools is selected on the Geometry ribbon tab:

- the Renumber Nodes tool in the Node group
- the Renumber Beams tool in the Structure group
- the Renumber Plates tool in the Plate group
- the Renumber Solids tool in the Solid group

![Renumber dialog](image)

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start number from</td>
<td>Specify the starting number of the renumbered entities. Depending on the New Numbering Order option selected, the new entity numbers will ascend or descend from this number.</td>
</tr>
<tr>
<td>New Numbering Order</td>
<td>Select either an Ascending or Descending order to control the subsequent entity numbers.</td>
</tr>
<tr>
<td>Available Sort Criteria</td>
<td>Sort criteria can be used to control the order in which selected entities will be numbered once the procedure is complete.</td>
</tr>
</tbody>
</table>
### List Operators

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected sort criteria to the Selected Sort Criteria list</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all sort criteria to the Selected Sort Criteria list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all entries in the Selected Sort Criteria list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected entry from the Selected Sort Criteria list.</td>
</tr>
</tbody>
</table>

### Selected Sort Criteria

Items added to this list will be used as criteria for sorting the numbered items. The order of operations is that the entities are renumbered starting with the lowest criteria on the list. Once this is complete for a criteria, the next higher criteria in the list will be used.

Use the Move Up or Move Down buttons to change the order of the items in the list.

**Note:** You can change the numbering order (ascending or descending) of individual sort criteria by double-clicking its entry in this list.

### Accept

Renumbers the selected model entities based on the specified parameters.

**Caution:** This operation cannot be undone.

### Cancel

Closes the dialog without renumbering any model entities.

### Help

Opens the STAAD.Pro help window.

### Related Links

- *M. To renumber selected beams* (on page 649)

### Define Member Attributes dialog

Used to pre-define member attributes such as properties, materials, beta angles, or release specifications for member attribute sets. Members which are then added to the model will have these properties by default.

Opens when the Set New Member Attributes tool is selected from the Add Beam drop-down in the Beam group on the Geometry ribbon tab.

#### Select Template

Contains a list of named templates, included user-defined templates created by clicking New.

#### New

Click to create a new template. Type a name and click OK.

#### Save

Save changes made to the currently selected template.

#### Rename

Click to rename the currently selected template.

#### Delete

Click to delete the currently selected template. You will be prompted to confirm this action.
Attributes group

Member Property
Select the Use Property option to predefine a Member profile. The Select Property: drop down list contains all the profiles currently being used in the STAAD.Pro command input file.

Beta Angle
Select the Use Beta Angle option to predefine a angle of rotation about the member's longitudinal axis. Options are available to use the Angle, the RAngle, or enter an Angle in Degrees.

Member Release
Select the Use Release at Start and/or Use Release at End options to predefine a member end release specification for either the start or end of the member(s). The Release at Start and Release at End drop-down lists contain all the member end releases currently being used at the respective member ends in the STAAD.Pro command input file.

Assign these attributes while creating new members
Select this option to ensure that these attributes will be assigned to any members which are created henceforth.

Related Links
- M. To set attributes for new beams (on page 643)

Insert Nodes into Beam # dialog

Used to insert one or more nodes at specified distances along selected members.

Opens when the Insert Node tool is selected in the Structure group on the Geometry ribbon tab.

One or more beams must be selected for this tool be active.

Beam Length
Lists the distance from node A to node B along the beam to be split.

New Insertion Point
Provide the Distance from the start node (as shown in the associated graphics) of the member in current length units. Alternatively, provide the Proportion of the total length of the member to position the new node. Click Add New Point to add the node.
Add New Point

Adds the node indicated by the **Distance** or **Proportion** value.

Add Mid Point

Splits the member into two equal segments.

Add n Points

To divide the beam in a number of equal segments, provide the number of intermediate points in the n = field and click **Add n Points**.

**Note:** This value should be an integer.

Insertion Points

A list of the locations of the newly created points is displayed here, shown as the distance from the start node of the member.

OK

Accept the added nodes and closed the dialog.

Cancel

Close the dialog without adding nodes.

Help

Opens the Help window.

Remove

To remove a node from the list of inserted nodes, highlight the desired node and click on this button.

Related Links

- *M. To insert a node in a single member* (on page 697)

**Insert Node / Nodes** dialog

Used to insert node(s) into multiple members simultaneously following the same prescribed method.

Opens when the **Insert Node** tool is selected in the Beam group on the Geometry ribbon tab (with multiple beams already selected).

**New point by distance**

Specify the distance in current length units at which the beam is to be split. The value for the distance is entered in the Distance edit box and is measured from the start node of the beam.

**New point by proportion**

This option allows the users to specify the distance in terms of a ratio. For example, to split a beam at the midpoint, enter 0.5 as the proportion. To split the beam at quarter points, use a proportion value of 0.25.

Add mid point

Splits the beams at their midpoints.

Add 'n' points

To split a beam by inserting n number of points, use this option. The beams are split up into n + 1 segments.

OK

Accept the added nodes and closed the dialog.

Cancel

Close the dialog without adding nodes.

Help

Opens the Help window.

Related Links

- *M. To insert a node in multiple members* (on page 697)
Stretch Member(s) dialog

Used to increase the length of a member in various ways.

Opens when the Stretch Members tool is selected in the Beams group on the Geometry ribbon tab.

Select member(s)

Multiple members can be selected simultaneously for stretching. However, whether they will get stretched or not depends upon the type of method used in stretching as described later. If you wish to remove one or more members from the list, uncheck the corresponding boxes within the Select member(s) drop-down list.

Select options

Select the method to use for stretching the selected members:

To a point

Specify the coordinates in current length units of the point to which one of the ends of the member are to be moved. The point must lie on the axis of the member being stretched, or else, the stretching will not be performed. The program automatically determines which of the two ends of the member is to be moved. So, this method involves

i. determining if the point lies on the axis of the member(s) being stretched

ii. determining which end to move for the member(s) which satisfy criteria (i)

iii. replacing the corresponding nodal coordinates with those of the desired point

Through a distance

Specify whether the start or end node will be stretched and the distance by which to stretch this end. In this method, all members selected for this operation will see the change in length.

You must create a node a resulting intersection point separately using the Intersect Beams tool.
To an existing node

This is very similar to the To a point method. The only difference is that instead of explicitly specifying the coordinates of the desired point, that point is already available for identification through its node number.

A list of all nodes in the model is available in the drop-down list. Alternately, click the Node selection tool to select a node graphically.

To an existing member

If two members lie within a plane, this method may be used to stretch the first member to meet the second member. The second member will be automatically split at the intersection point into two segments.

A list of all members in the model is available in the drop-down list. Alternately, click the Member selection tool to select a member graphically.

Details

This button toggles the display of the detailed actions taken during the stretch operation.

Tip: If this button is disabled, then the selected stretch operation is invalid.

Related Links

• M. To stretch a member (on page 648)

Merge Selected Beams dialog

Used to join two collinear beams and replace them with one beam. The intermediate node(s) (common node) is removed from selected members and the incidences are redefined.

Opens when the Merge Selected Beams tool is selected in the Beam group on the Geometry ribbon tab.

Beam No

Select the beam no to use for the merged beam.

Section Profile

Select the section property to use for the merged beam.
Beta Angle
  Select the local x axis rotation value to use for the merged beam.

Material
  Select the material definition to use for the merged beam. Material data is listed below.

**Note:** If the members are defined using material constants instead of material definitions, then the individual constants will be available to select for the merged beam instead.

**Related Links**
- *M. To merge two or more members* (on page 648)

**Parametric Models** dialog

Used to automatically generate finite element meshing for defined boundary areas.

Opens when the Parametric Models tool is selected in the Plate group on the Geometry ribbon tab.

**Preview Models list**
  A list of mesh prototypes are listed here. Multiple settings may be experimented with here to obtain the desired surface mesh before merging with the base model.

- **Boundary Connectivity**

  Selecting a Boundary Point displays the point's 3D coordinate which cannot be modified and two additional parameters which can:

  - **Division.** When a Mesh Type B) has been selected, this value is used to identify the number of divisions along the boundary edge from the selected node to the next defined point on the boundary.
  - **Density.** When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the boundary point.

- **Openings**

  Openings can be added by selecting the Openings heading and clicking the “Add” button. Either polygonal or circular openings can be added as described above. Both opening types have options that can be edited.

  - **Polygonal**

    Selecting a Polygonal opening definition in the dialog displays the coordinates that form the opening (which cannot be edited). Each point includes two additional parameters which can be edited:

    **Division.** When a Mesh Type B) has been selected, this value is used to identify the number of divisions along the boundary edge from the selected node to the next defined point on the boundary.
    Density. When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the opening point.

  - **Circular**

    Selecting a Circular opening definition in the dialog displays the coordinates that form the opening (which cannot be edited). Each point includes three additional parameters which can be edited:

    **Radius.** This defines the size of the opening. This value is used by all Mesh Types.
Division. This is used to define the number of segments that form the circle and is used by all Mesh Types.

Density. When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around each point that forms the opening.

- **Regions**
  Selecting a Region definition in the dialog displays the coordinates that form the region (which cannot be edited). Each point includes two additional parameters which can be edited:
  - Division. When a Mesh Type B) has been selected, this value is used to identify the number of divisions along the boundary edge from the selected node to the next defined point on the boundary.
  - Density. When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the point on the region.

- **Density Points**
  Selecting a Density Point definition in the dialog displays the coordinate of that point, which cannot be edited and a single parameters which can be edited:
  - Density. When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the density point.

- **Density Lines**
  Density lines are defined by two points. Selecting a Density Line definition in the dialog displays the two coordinates of the line which cannot be edited and two additional parameters which can:
  - Division. This is displayed on the first point that defines the density line, but applies to the line between both points. This value is used to identify the number of divisions along the density line and is used by all mesh types.
  - Density. When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the point on the density line.

**Parameters table**
Displays the parameters for the selected mesh model, Opening, Region, Density Point, or Density Line. Changes made here can be saved using the **Apply** button.

- Density - A measure of the fineness to which the region around the node has been meshed. To obtain a more refined mesh around a node, increase the density value in the row corresponding to that node.

**Add...**
Used to add a new mesh model to the list or to add a new Opening, Region, Density Point, or Density Line to the current mesh model.

**Delete**
Removes the selected mesh model from the Preview Models list or removes the selected Opening, Region, Density Point, or Density Line from the current mesh model.

**Caution:** No confirmation is used to delete mesh models or their sub-components. This action cannot be undone.

**Apply**
Updates the selected mesh model with any changes made to the settings.
**Merge Mesh**
Commits the selected mesh model to the STAAD.Pro input file base model.

**Caution:** Until this button has been clicked for one or more mesh models, no data from this page will be saved.

**Related Links**
- *AD.2006.1.3 Persistency of Parametric Mesh Model in STAAD Input File* (on page 321)
- *TR.14.1 Parametric Mesh Models* (on page 2434)
- *M. To create a parametric surface model* (on page 657)

**New Mesh Model dialog**
Used to name a new mesh model.
Opens when **Add** is clicked to create a new mesh model in the **Parametric Models** dialog (on page 2910).

<table>
<thead>
<tr>
<th>Name</th>
<th>Enter a unique name here to identify the mesh model in the Parametric Models dialog Preview Models list.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Use nodes and beams that occur on or inside the outer boundary as additional density points and lines?</strong></td>
<td>Select this option to have the program auto-detect points on the outer boundary not explicitly clicked when defining the edge of the mesh boundary. The program will also use any beams and nodes inside the boundary as density lines and points, respectively.</td>
</tr>
<tr>
<td><strong>OK</strong></td>
<td>Closes the dialog and allows you to begin defining points around the mesh boundary.</td>
</tr>
<tr>
<td><strong>Cancel</strong></td>
<td>Closes the dialog without creating a new mesh model.</td>
</tr>
</tbody>
</table>

**Related Links**
- *M. To create a parametric surface model* (on page 657)

**Mesh Parameters dialog**
Used to specify the parameters used for automatically generating a finite element mesh over boundary area.
Opens when a new mesh boundary has been defined from the **Parametric Models** dialog (on page 2910).

**Note:** The meshing features described here requires STAAD.Pro 2007 (build 03) and higher.
Meshing Method

Select one of the following mesh methods:

- **Basic** - Can make use of any of the three **Boundary Connectivity** (meshing type) options but only supports Triangular elements.
- **Standard** - Can use type B or C **Boundary Connectivity** (meshing type) and only supports Quadrilateral elements.

Element Type

Select either to use **Triangular** only or predominantly **Quadrilateral** elements.

- **Triangular** - Only triangular elements will be created.
- **Quadrilateral** - This will create principally four sided elements, but where the geometry dictates, some places may require triangular elements.

Boundary Connectivity

A. **Only use selected boundary nodes, boundary edges not divided** - Meshing will be done without creating any nodes along the edges of the boundary we defined. Thus, the entire distance between 2 sequential nodes will form the side of one triangular element. In most cases, this can lead to poor quality elements, and is best avoided.

B. **Divide boundary segments with specified divisions** -

C. **Divide boundary segments to optimize mesh quality** - (Default, recommended)

Automatic boundary divisions.

The figures in the dialog correspond to the appropriate options.
Target element size

Specify a target dimension to use as a guideline when automatically generating the mesh. This value is nominally used when dividing the boundary region into finite elements, but the element size is also determined by other factors such as the distance between boundary and other density points.

Default Number of Divisions of Boundary Edges

Specify the number of boundary edges to use along a boundary segment.

OK

Closes the dialog and generates the mesh. If you are using the Geometry | Parametric Model page, a dialog opens asking if you wish to add openings to the mesh.

Cancel

Closes the dialog without generating a meshed surface.

Related Links

- M. To create a parametric surface model (on page 657)

Define Meshing Region dialog

Used to specify data such as the number of divisions along each side of the polygon, and whether any cutouts or holes are to be specified in the polygon.

Opens when a polygonal meshing is being defined.

Polygonal Plate tree

Entries in the tree represent the Boundary (polygonal surface edges) and the included Holes, if any.

Selecting the HOLES entry allows you to add holes or delete all holes either from the right-click pop-up menu or using the HOLES tools:

Table 288: HOLES tools in the Define Meshing Region dialog

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Add New Hole]</td>
<td>Adds a new hole for the current boundary region. Coordinates for the hole corners must be defined.</td>
</tr>
<tr>
<td>![Delete All Holes]</td>
<td>Deletes all holes currently added in the boundary region.</td>
</tr>
</tbody>
</table>

Table

Each row in the table represents a corner node of the selected Boundary or Hole, along with the number of edge divisions and bias for the edge between the node on this row and the previous row.

- X / Y / Z — Global coordinates of the Boundary or Hole corner.
- Div. — Number of elements will be created between the current node represented by the current row and that of the previous row (division).
• Bias — Used to control the variation in division length along the edge. Represents the ratio of length of the last element to the first element along a side. That is, the last element will be (bias value) times the length of the first element. A value of unity (1) results in equal divisions along a side.

Table 289: Table tools in the Define Meshing Region dialog

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Add New Row" /></td>
<td>Adds a new row to the bottom of the table with default values.</td>
</tr>
<tr>
<td><img src="image" alt="Delete Row" /></td>
<td>Removes the currently selected row from the table.</td>
</tr>
<tr>
<td><img src="image" alt="Rename Hole" /></td>
<td>(When hole is selected) Used to provide a user-defined name for the hole.</td>
</tr>
<tr>
<td><img src="image" alt="Delete Hole" /></td>
<td>(When hole is selected) Removes the current selected hole from the polygonal plate tree.</td>
</tr>
</tbody>
</table>

**OK** Generates a meshed surface with the specified parameters.

**Cancel** Closes the dialog without generating a meshed surface.

**Reset** Undoes any changes made to the dialog.

**Select Meshing Parameters** dialog

Used to name and specify parameters for a quadrilateral plate super-element.

Opens when a quadrilateral meshing is selected.

- **Model Name** Specify an optional, user-defined name for the quadrilateral super-element.

- **Corners** The X, Y, and Z coordinates of each corner (the selected nodes) are labeled A through D.

- **Length, Bias & Division** For each side connecting two Corners, the straight line distance is provided. You may also edit the Bias and Divn. values for each side to control the meshing.

  - **Div.** — Number of elements will be created between the current node represented by the current row and that of the previous row (division).
  - **Bias** — Used to control the variation in division length along the edge. Represents the ratio of length of the last element to the first element along a side. That is, the last element...
Element Type  Select either a Triangle or Quadrilateral element for the finite elements which compose the super-element mesh.

Apply  Closes the dialog and generates a quadrilateral mesh with the specified parameters.

Cancel  Closes the dialog without creating a quadrilateral mesh.

**Define Plate Object Property** dialog

Used to define various attributes of the plate to be pre-defined.

Opens when the **Add Plate > Set New Plate Attributes** tool is selected in the **Plate** group on the **Geometry** ribbon tab.

- **Use Property**  Select the Use Property option to predefine a Plate thickness. The **Select Property** drop down list contains all the thicknesses currently being used in the STAAD.Pro command input file. Additional thicknesses can be added using the Create New Property button.

- **Create New Property**  Opens the **Plate Element/ Surface Property** dialog to the Plate Element Thickness tab, which is used to define the thickness of the plate at each corner node.

- **Use Material**  Select the Use Material option to predefine a plate Material. The **Select Material** drop down list contains all the materials currently being used in the STAAD.Pro command input file. Additional materials can be added using the the Create New Material button.

- **Create New Material**  Opens the **Isotropic Material** dialog, which is used to define Material properties for a new isotropic material type or select a predefined, common isotropic material.

- **Use Release**  Select the Use Release option to predefine a set of plate releases. The **Plate Release** drop-down list contains all the plate releases currently being used in the STAAD.Pro command input file. Additional releases can be added using the Create New Releases button.

- **Create New Release**  Opens the Plate Specs dialog, which is used to define the degrees of freedom to be released at each node of the plate as well as define the plate to be plane stress, no in-plane rotation, or no stiffness.

- **Assign these attributes while creating new plates**  Select this option for the program to recognize the pre-defined attributes. Any new plate element created from here on (whether created individually, through a mesh or Structure Wizard) will now possess these attributes.

**Related Links**
- *M. To set new plate attributes* (on page 653)

**Composite Deck** dialog

Used to create and define composite decks.

Opens when the **Composite Deck Layout** tool is selected in the **Composite Deck** group on the **Geometry** ribbon tab.
All composite deck groups and their constituent members are listed here.

**New Deck options**

- Clicking on Nodes - Used to define the deck by clicking on the nodal points that comprise the deck boundary. Click on the **Create New Deck** button to begin selecting the deck boundary using the composite boundary cursor. Once the deck is created, it will be shown with a hatch pattern.

- Use Selected Beams - Used to define the deck by selecting the beams that comprise the deck boundary. Using the Beams cursor, select the beams which will define the outer boundary of the deck.

**Create New Deck**

Used to define the deck by the selected method. Once the deck is identified by any one of the aforementioned methods, a dialog box as shown below will prompt for the deck name. Enter an appropriate deck name in the edit box provided.

The deck will then be defined and the deck name will appear in the **Composite Deck** box as shown below.

Select the deck in the tree control to define the properties of the deck. The various options in the **Composite Deck** box are explained as follows.

**Create Direction**

Used to define the deck direction (direction of the longitudinal flutes or ribs in the deck) by highlighting any two beams and clicking the **Create Direction** button. The deck direction will be defined to run perpendicular to the two beams that were selected. A symbol will appear to indicate the direction of the deck. This symbol will be placed at the centroid of the deck area.

Once the direction of the deck is specified, the beam numbers which comprise the deck will be displayed as child elements of the deck group.

Click the color square (adjacent to the Create Direction button) to open a color picker dialog. Select a custom color to hash the selected deck or any newly created decks. Each deck can have a different color.

**Tip:** Use different colors to quickly identify composite decks with different properties or directions.

**Deck Properties**

The following options appear in the Composite Deck dialog when a deck group is selected.

**Update Deck Property**

Synchronizes the graphical window and updates any changes made to the deck definition.
Concrete Properties
Specify the concrete thickness above the flutes, the unit weight of concrete and the grade of concrete in the respective edit boxes.

Rib Properties
Rib properties can now be defined either using the database or by entering the properties manually.

Use Database
A drop-down list of the various standards incorporated in STAAD.Pro. The program includes three standard catalogs: ASC™, Vulcraft™, and VERCO™. The rib properties will be used from the database provided.

Apart from using the available standards, you may also define custom rib properties by providing the Rib Width and Rib Height.

Rib Width
Width of the rib in this box. If a database deck is selected, this value is updated.

Rib Height
The height of the rib in this box. If a database deck is selected, this value is updated.

Stud Diameter
Specify a diameter for the Stud by choosing the Custom option or select from the three available diameters: 3/4", 5/8", 1/2".

Stud Length
Specify the length (height above deck pan) of the stud in this box.

Shored
Choose the appropriate radio button depending on whether or not shoring is to be used during construction.

Beam Properties
The following options appear in the Composite Deck dialog when a component beam is selected.

Current Property
The current section and material of the selected beam is displayed.

Add / Change Property
Opens the appropriate Steel Table dialog (on page 2969) which is used to select a steel section and material.

Effective Width
The effective width for each beam is automatically calculated by STAAD.Pro and is displayed in the Effective Width box. The user can modify the effective width for any individual beam by inputting the value in the Effective Width box and clicking on the update button.

Related Links
- M. To create a new composite deck from perimeter beams (on page 694)
- M. To specify a direction for the composite deck ribs (on page 694)
- M. To assign composite deck properties (on page 695)
- M. To modify composite steel beam properties (on page 695)
- M. Composite Decks (on page 693)
- G.6.7 Composite Beams and Composite Decks (on page 2328)
- M. To add a floor load or one-way load (on page 831)
<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Labels Settings</td>
<td>Opens the <strong>Diagrams</strong> dialog to the <strong>Labels</strong> tab, which is used to customize the view of the structure by setting different view-related parameters.</td>
<td></td>
</tr>
<tr>
<td>Node Labels</td>
<td>Select any of the label types from the drop-down list to turn the display of that label on or off.</td>
<td>&lt;Shift+N&gt;</td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> Additional labels are controlled on the Diagrams dialog <strong>Labels</strong> tab.</td>
<td></td>
</tr>
<tr>
<td>Beam Labels</td>
<td></td>
<td>&lt;Shift+B&gt;</td>
</tr>
<tr>
<td>Plate Labels</td>
<td></td>
<td>&lt;Shift+P&gt;</td>
</tr>
<tr>
<td>Solid Labels</td>
<td></td>
<td>&lt;Shift+C&gt;</td>
</tr>
<tr>
<td>Individual Node Labels</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Individual Beam Labels</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
<td>----------</td>
</tr>
<tr>
<td><strong>Individual Plate Labels</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Individual Solid Labels</strong></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 291: Tools group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Zoom Window</strong></td>
<td>Used to define the boundaries of a rectangular area of the active view to be displayed within the current view.</td>
<td></td>
</tr>
<tr>
<td><strong>Whole Structure</strong></td>
<td>Fits the current view to display the entire structure limits. Resets view rotation to orthogonal.</td>
<td></td>
</tr>
<tr>
<td><strong>Zoom In</strong></td>
<td>Zoom in the current view by a preset percentage magnification.</td>
<td>scroll wheel up</td>
</tr>
<tr>
<td><strong>Zoom Out</strong></td>
<td>Zoom out the current view by a preset percentage magnification.</td>
<td>scroll wheel down</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>----------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Pan</td>
<td>Used to view a different part of the design without changing the view magnification.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Tip: Press and hold the middle mouse button (typically the scroll wheel button) to quickly pan in the view window.</td>
<td></td>
</tr>
<tr>
<td>Zoom Extents</td>
<td>Adjusts the view magnification so that the entire model is visible in the view.</td>
<td></td>
</tr>
<tr>
<td>Zoom Factor</td>
<td>Opens the Zoom Factor dialog, which is used to zoom in or out of the structure by specifying a magnification factor. Factors less than one Zoom out.</td>
<td></td>
</tr>
<tr>
<td>Dynamic Zoom</td>
<td>Used to define the boundaries of a rectangular area of the active view to be displayed within a new view. The previous view window highlights the location of the zoomed portion.</td>
<td></td>
</tr>
<tr>
<td>Magnifying Glass</td>
<td>Provides a magnified portion of the current view window when you click and drag the pointer.</td>
<td></td>
</tr>
<tr>
<td>Isometric View</td>
<td>(Default view) View the model as an isometric projection.</td>
<td></td>
</tr>
<tr>
<td>Front View</td>
<td>View the structure model from the positive Z axis.</td>
<td></td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>----------------</td>
<td>------------------------------------------------------------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>Left View</td>
<td>View the structure model from the negative X axis.</td>
<td></td>
</tr>
<tr>
<td>Top View</td>
<td>View the foundation mode in plan; from the positive Y axis.</td>
<td></td>
</tr>
<tr>
<td>Back View</td>
<td>View the structure model from the negative Z axis.</td>
<td></td>
</tr>
<tr>
<td>Right View</td>
<td>View the structure model from the positive X axis.</td>
<td></td>
</tr>
<tr>
<td>Bottom View</td>
<td>View the structure model from the negative Y axis.</td>
<td></td>
</tr>
<tr>
<td>Rotate Up</td>
<td>Rotate the structure model forward about the X axis.</td>
<td>↑↓ (Up arrow key)</td>
</tr>
<tr>
<td>Rotate Left</td>
<td>Rotate the structure model forward about the Y axis.</td>
<td>←→ (Left arrow key)</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
<td>Shortcut</td>
</tr>
<tr>
<td>------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>---------------------</td>
</tr>
<tr>
<td>Spin Left</td>
<td>Rotate the structure model forward about the Z axis.</td>
<td>&lt;Ctrl+←&gt; (Ctrl+Left arrow keys)</td>
</tr>
<tr>
<td>Rotate Down</td>
<td>Rotate the foundation mode backward about the X axis.</td>
<td>↓ (Down arrow key)</td>
</tr>
<tr>
<td>Rotate Right</td>
<td>Rotate the structure model backward about the Y axis.</td>
<td>←→ (Right arrow key)</td>
</tr>
<tr>
<td>Spin Right</td>
<td>Rotate the structure model backward about the Z axis.</td>
<td>&lt;Ctrl+→&gt; (Ctrl+Right arrow keys)</td>
</tr>
<tr>
<td>Toggle View Rotation Mode</td>
<td>Used to select an node as the center of rotation. When toggled on, pressing &lt;Ctrl+Shift&gt; and clicking a node will set that node as the center of rotation.</td>
<td></td>
</tr>
<tr>
<td>Orientation</td>
<td>Opens the <strong>Orientation</strong> dialog, which is used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.</td>
<td>&lt;F4&gt;</td>
</tr>
</tbody>
</table>
### Always Fit in Current Window

Instructs the program on what guidelines to use when drawing a selected set of objects on the screen. It displays the selected portion of the model to a size governed by optimum usage of the dimensions of the current window.

When you select various tools the program or switch workflows the size of the drawing window frequently changes. With this tool turned on, the model or selected portions of it, will be drawn in such a manner that all the entities will be drawn within the bounds of the drawing area. This means, the size to which the entities are drawn will correspondingly increase or decrease.

With this tool turned off, the size of the entities will remain constant, but that means it may or may not fit within the bounds of the drawing window.

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Always Fit in Current Window</strong></td>
<td>Instructs the program on what guidelines to use when drawing a selected set of objects on the screen. It displays the selected portion of the model to a size governed by optimum usage of the dimensions of the current window. When you select various tools the program or switch workflows the size of the drawing window frequently changes. With this tool turned on, the model or selected portions of it, will be drawn in such a manner that all the entities will be drawn within the bounds of the drawing area. This means, the size to which the entities are drawn will correspondingly increase or decrease. With this tool turned off, the size of the entities will remain constant, but that means it may or may not fit within the bounds of the drawing window.</td>
</tr>
</tbody>
</table>

Table 292: Views group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Open View</strong></td>
<td>Opens the Open View dialog, which is used to open a previously saved structural view window.</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro User Manual**
### New View

Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way.

**Note:** This option becomes active only after you select one or more structural elements on screen.

### Selected Objects

Hides all objects that are not part of the current selection.

### View Management

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Detach View</strong></td>
<td>Used to delete any saved or active view window.</td>
</tr>
<tr>
<td><strong>Add to View</strong></td>
<td>Used to add geometry to any saved or active view window.</td>
</tr>
<tr>
<td><strong>Save View</strong></td>
<td>Opens the <strong>Save View As</strong> dialog, which is used to save the current view of the structure with a name. Type a name for the view. This view may be opened later for performing verification/visualization operations.</td>
</tr>
<tr>
<td><strong>Rename View</strong></td>
<td>Opens the <strong>Rename View</strong> dialog, which is used to rename an existing view. Type a <strong>New Name</strong> for the view. The old name is shown for reference.</td>
</tr>
</tbody>
</table>

### Display Options

Opens the **Options** dialog, which is used to change the length and force units for various values which can be displayed in the graphics window, such as member properties, material constants, load magnitudes, plate stresses, etc.
### Tool name | Description
---|---
Set Structure Colors | Opens the **Color Manager** dialog, which is used to set colors for highlight, entities, and results.
Structural Tooltip Options | Opens the **Tool Tip Options** dialog, which is used to set tooltip options for structural elements.

**Table 294: Windows group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cascade</td>
<td>Stacks view windows in order with the active window entirely visible and the title bar of each remaining window visible.</td>
<td>&lt;Shift+F5&gt;</td>
</tr>
<tr>
<td>Tile Horizontal</td>
<td>Tiles view windows vertically, with each window in a landscape (horizontal layout). This is a quick way to clean up the screen.</td>
<td>&lt;Shift+F4&gt;</td>
</tr>
<tr>
<td>Tile Vertical</td>
<td>Tiles view windows horizontally, with each window in a portrait (vertical layout). This is a quick way to clean up the screen.</td>
<td>&lt;Ctrl+Shift+F4&gt;</td>
</tr>
<tr>
<td>Structure Only</td>
<td>Closes all open view windows, non-modal dialogs, and tables and opens a View window of the full structure expanded to the full program window.</td>
<td></td>
</tr>
<tr>
<td>Window</td>
<td>A list of all open and saved Views is displayed here. Select any number to make this the active view.</td>
<td></td>
</tr>
</tbody>
</table>
### Diagrams dialog

Used to customize the view of the structure by setting different view-related parameters.

Opens when either:

- the **Labels** tool is selected in the **Labels** group on the **View** ribbon tab

- when either **Labels** or **Structure Diagrams** is selected from the right-click pop-up menu in the View area
Structure tab

Used to set up structural view parameters.

**3D Sections**

The options in this group control how the members are displayed (only one may be selected).

- **None** - Displays the structure without displaying the cross-sectional properties of the members and elements.
- **Full Sections** - Displays the 3D cross-sections of members, depending on the member properties.
- **Sections Outline** - Displays only the outline of the cross-sections of members.

**View**

The options in this group allow additional view-related operations on the structure.

- **Fill Plates/Solids/Surface**
  
  Fills up the plate and solid and surface elements, if present.

- **Hide Plates/Solids**
  
  Hides all plate and solid elements from the view.

- **Hide Structure**
  
  Hides the entire structure from view. This option may be used to switch off the original structure view while displaying the deflected shape of the structure or the mode shapes.

- **Show Center Lines**
  
  Displays the centerlines of the members.

- **Shrink**
  
  Displays the individual structural elements detached from each other and helps to view their connectivity. The individual members are not drawn to full length or full width but shrunk by a percentage provided in the associated edit box.

- **Perspective**
  
  Change current view to perspective.

- **Sort Geometry**
  
  Changes the display of plate and solid element results so that different elements are in front.

- **Sort Nodes**
  
  Changes the display of node results so that different nodes are in front.

- **Draw Deck Hatch**
  
  Hatches in composite deck areas.

**Colors**

Allows you to display and change the color of beams, plates, solids, section outline and selected entities.

- **Beams**
  
  Click Default to select the color used for beams.

- **Plates**
  
  Click Default to select the color used for plates.
Plate elements can be color coded to distinguish the directions of their local z-axes. The face of the element in the positive direction of the z-axis is referred to as the “front.” The opposite side is referred to as the “back.”

**Solids**
- Click Default to select the color used for solids.

**Section Outline**
- Click to select the color for the **Sections Outline** option in the 3D Sections group.

**Selected Entities**
- Click to select the color of selected entities in the View area.

**Show Color Legend**
- Selecting this option will display a color legend on the screen.

**Margin around Structure**
- Represents the blank margin around the structure in percentage of the total view window.

**Solid – Fill Face**
- Select a face of the solid to be filled with color. After checking this option and clicking **Apply**, the selected face of solids are filled with color.

**Load and Results tab**
- Used to specify the scales to which the different diagrams for viewing input and results should be plotted.

**Scales tab**
- Used to specify the scales to which the different diagrams for viewing input and results should be plotted.

**Tip:** In general, a larger scale number causes the diagram to shrink and a smaller scale number enlarges the diagram.

**Save as Default**
- If the scale, to which an input item or result item is drawn, has been modified, one may instruct the program to keep that setting as the default, in place of the previous setting or the built-in default.

**Reset to Default**
- Resets all scales to the installed default values.

**Apply Immediately**
- When checked scale values in the view window are updated immediately as you change the value.

**Icon Size**
- **Support Icon** The size to which the fixed-but support icons are plotted can be increased or decreased using this option. This applies to fixed but support icons only.
- **Master Block** Controls the size of the master node icon when the **Master Slave** option has been checked on the **Labels** tab.

**Labels tab**
- Used to select various display labels for different components of the structure.

**Nodes**
- **Node Numbers** displays the node numbers on screen.
**Node Points** identifies the nodes with a small circle.

**Supports** displays the support icons at supported nodes.

**Dimension** displays the member lengths in current units.

**Properties**

- **References** displays the Property Tag number of the member/element properties.
- **Sections** displays the section name (such as W12x26).
- **None** removes the display of property information.

**Physical Members**

These options work only in the steel design mode. Please refer to the Miscellaneous section at the end of this manual for more details.

**Beams**

- **Beam Numbers** displays the member numbers on the frame members.
- **Beam Orientation** displays an icon showing the local axis of each member. The arrow indicates the positive direction of the local x-axis. The local y-axis is in the direction of the thicker flange. Please note that the I symbol is used regardless of the actual section type.

  **Note:** The local x-axis is represented by the blue arrow. The local y-axis is represented by the red arrow. The local z-axis is represented by the green arrow.

- **Beam Spec** option displays beam specifications which have been assigned, such as truss and tension only members.
- **Releases** displays the member releases.
- **Beam Ends** This switch, if turned on, paints the beams in the current view, with distinct colors to distinguish the start node of the member from its end node, thereby making it easy to identify the direction of the local X axis of the member.
- **Start Color** Click on the little square adjacent to this option to set the color in which the zone of the member adjacent to its start node must be painted in.
- **End Color** Click on the little square adjacent to this option to set the color in which the zone of the member adjacent to its end node must be painted in.

**Plates**

- **Plate Numbers** displays the plate element numbers.
- **Plate Orientation** displays the local axis system for plates.

**Solids**

- **Solid Numbers** displays the solid element numbers.

**Surfaces**

- **Surface Numbers** displays the surface numbers
- **Surface Orientation** displays the local axis system for surface object

**Loading Display Options**

- **Load Values** Check this box to display the intensity of the loads in the current unit system.
### Display Floor Loading
Check this box to enable/disable the display of the actual joint/uniform load icons applied to the nodes or members.

### Display Floor Load Distribution
Check this box to display the tributary area diagrams for floor loads.

### Display Wind Load Contributory Area
Check this box to display the tributary area diagrams for wind loads.

### Display Wind Load
Check this box to enable/disable the display of the actual joint/uniform load icons applied to the nodes or members.

### General

**Show Axes Window**
Check this option to display a small icon at the bottom left corner of the drawing area which displays the directions of the global X, Y and Z axes are drawn with respect to the current view of the structure.

**Show Axes at Origin**
Check this option to display an icon of the global axes system at the global X=0, Y=0, Z=0 position on the drawing.

**Material**
Displays the name of any material assigned.

**Show Diagram Info**
Through the various pages of the post processing mode, the program provides the facility for plotting diagrams such as the Node displacements, beam moments, plate stress contours, etc. Along with these diagrams, a caption is displayed along the bottom right corner of the drawing window indicating the type of diagram, load case, degree of freedom, length and force units, etc. In order to bring up those captions, this switch must be turned on. If you do not want those captions to come up, switch off this label.

**Show Master Slave**
Set this option to display master joints as green cubes.

### Always Use STAAD.Pro Default Label Settings
As you navigate the various pages of the program, certain labels—meant for easy identification of the assigned attributes—will become switched on by default. Keeping this setting on enables that.

### Always Use Current Label Settings
If you do not wish to have the default labels of each page come on by default (see the item above), one may override that setting and switch on only the labels one chooses to display on a page by page basis. Switch this setting on to override the default labels display. By page, we are referring to the various tabs that appear along the left side of the screen.

### Use Partial Labeling Mode (use labeling cursors to turn ON/OFF individual labels)
If we switch on a label, such as say, node numbers, or beam numbers, by default, STAAD.Pro will display those numbers for all the nodes and beams which appear in the current view. On large models, or if several types of labels are turned on, this can clutter the drawing making it difficult to distinguish the individual entities. The **Partial Labeling Mode** enables us to switch on labels for only a select few nodes, members etc. This switch must be used in conjunction with the partial labeling cursors.

### Force Limits tab
Used to identify the members having the force values lying inside or outside the specified ranges.
### Load Case
Select the **Load Case** for which you want to display the diagram.

### Forces
Select the force type from the list of available options, such as Axial, Shear YY, Bending ZZ, etc. Provide the range in the associated edit box under **Minimum** and **Maximum**.

### View Limits
The buttons under this group determine which members are going to be highlighted. The **Exceed Either** option highlights any member whose forces exceed either the **Maximum** or the **Minimum** values. **Exceed Maximum** option highlights only the members whose forces exceed the **Maximum** values. **Exceed Minimum** option highlights only the members whose forces exceed the **Minimum** values.

### Animation tab
Used to graphically view deflections, section displacements, mode shapes and stresses in an animated mode. It is a way of visualizing the stress build-up or attainment of peak displacement using animated views.

<table>
<thead>
<tr>
<th>Diagram Type</th>
<th>No Animation - select this option to turn off any current animation</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Deflections</td>
</tr>
<tr>
<td></td>
<td>Section Displacements</td>
</tr>
<tr>
<td></td>
<td>Mode</td>
</tr>
<tr>
<td></td>
<td>Stress</td>
</tr>
</tbody>
</table>

#### Animation Setup
- **Full Screen**
  - displays the animation in the full monitor screen rather than in a window. This option may use less memory than displaying the animation in a large window.
- **Extra Frames**
  - Choose the number of **Extra Frames** above the minimum needed to enhance the animation if it appears choppy.
- **Target FPS**
  - Choose the **Target FPS** (frames per second) to control the animation speed. To speed up an animation, show more frames per second. To slow down an animation, show fewer frames per second.
- **Use Metafiles for offscreen data**
  - Click the **Use Metafiles for offscreen data** to save the animated screens as Windows Metafiles. Speed may be slower if this option is used.

### Design Results tab
Used to display code check and steel design information on the structure.

- **Show Diagram options**
  - **None** - Do not display the design results graphically
  - **Show Diagram (Based on Actual Ratio)** - Displays the design results diagram in the active view window. Values and colors displayed are based on the ratio of the demand to capacity ratio for the selected load case.
• **Show Diagram (Based on Normalized Ratio)** - Displays the design results diagram in the active view window.

• **Basic Diagram** - Members are displayed in four distinct colors which indicate Not Designed, Pass or Safe, Fail and Extreme Failure respectively. The default values define a member as Safe if the ratio is less than 1, Fail if the ratio is between 1 and 1.5, and Extreme Fail if the ratio is greater than 1.5.

  **Note:** It is important to understand that in this page, the definition of Pass and Fail can be set to have a different meaning than the one the calculation engine used during the steel design process. In the calculation engine, Fail means exceedance of the value set for the RATIO parameter (whose default value is 1.0 usually).

• **Detailed Diagram** - The table on the right hand side of the box displays the index for the color plot. The first column indicates the lower limit and the second column represents the upper limit for the unity ratio. You can accept the default index settings or can choose to define your own settings by selecting the Use Custom Limits option. The upper and lower limit for the ratio can be provided in the Maximum and Minimum fields, respectively. Users can control the number of divisions between the maximum and minimum values for the index through the No of values option. The values for the divisions or ranges can also be set by the user by checking the Use Custom Divisions check box which will allow the user to edit the ranges displayed in the table on the right.

**Show values** Select this option to display the ratio values on the contour plot.

**Actual Ratio ranges table** Displays the From and To range used for a color to graphically represent the design results status of a member. By default, members passing all design checks performed will be shown in green, members failing design checks by 50% or less shown in blue, and those failing by greater than 50% shown in red.

**Tip:** Double click the colored index cell to open a **Color** dialog, which is used to set a custom color for the selected range.

**Plate Stress Contour tab**

Used to display stress contours for plates for different types of stresses.

**Tip:** This tab is only present when plate elements are in the model.

**Load Case** Select the Load Case for which the stress is to be displayed.

**Stress Type** Select the type of stress gradients to display on the elements:

- Max Absolute
- Max Top, Min Top, Max Bottom, Min Bottom (Principal Major and Minor Stresses)
- Tau Max Top, Tau Max Bottom
- Max Von Mis
- Von Mis Top, Von Mis Bottom
Max Tresca
Tresca Top, Tresca Bottom
SX, SY, SXY, MX, MY, MXY, SQX, SQY (local)
Global Moment, Global Stress
Base Pressure
Top Combined SX, Top Combined SY, Top Combined SXY, Bottom Combined SX, Bottom Combined SY, Bottom Combined SXY (local)

Contour Type
The Normal Fill option provides contours based on the values of centre stress and corner stress as described above.

The Enhanced Fill option will determine the contour using the centre and corner stresses as used in the Normal Fill option, and an additional set of interpolated control points along the edges of each plate that will tend to smooth out the contour lines on plates that have large changes of stress over their surface.

Absolute Values
causes the stress values to be compared based on the absolute values, rather than algebraic values. If this option is checked, stress values of +10 units and -10 units will be in the same range.

Index Based on Center Stress
The Plate Stress Contour maps are based on values from the calculated analysis results of the centre stress/force of each plate and corner stresses (which are averaged from all plates that share the same node). There are two ways that the corner stresses/forces can be determined. This can be either directly from the analysis results or estimated by extrapolating the centre stresses to the corners of each plate. These method to use is specified with the option:

• When checked, the contour map will be drawn using only the values of centre stresses/forces calculated by the analysis. Values for corner stresses/forces will be determined from each centre stress by extrapolating them to the corner and then averaged for all plates that share the node.
• When unchecked, the contour map will be based on both the centre stresses/forces AND the values of corner stress/force calculated by the analysis. Again the value is averaged for all plates that share the node.

View Stress Index
displays the legends of the colors with stress values at the side of the screen.

Re-Index for new view
When individual elements are isolated from the full model then the stress distribution for the entire element may be displayed in a single color. This is because the variation in stresses across the element may not have been appreciable with respect to the entire model and the entire stress range may have been represented by a single color in the full model. This may create problem because the users in such cases will not be able to view the stress variation across the element. To take care of this The re-index stress range for new view option has been included which automatically regenerates the stress index for any newly created view which will allow the users to view the stress variation in much more detail.

Show Displaced Shape
Displays the plate stresses on a displaced model.
### Contour Base on Visible Entities Only

Limits the stress contour range to the stresses contained in only visible entities.

### Use Custom Limits

By default, 15 contours are plotted with the maximum and minimum values determined by the program based on the results for the selected load case and stress type.

Set the Use Custom Limits option to specify **Minimum** and **Maximum** values as well as **No. of values** to use for equal divisions.

### Use Custom Divisions

By default, the values of the stresses used for color gradation are obtained by taking the **Maximum** and **Minimum** and dividing the difference equally into as many parts as is defined **No. of values**.

Setting the Use Custom Divisions allows you to directly type the cut-off values for each color.

**Tip:** Take care to ensure that they are in ascending order.

### Directions for Global Stress

(Global Stress or Global Moment only) Select the **Result Dir** of interest and the vector direction for **Up**.

### Solid Stress Contour tab

Used to display stress contours for solids for different types of stresses.

**Tip:** This tab is only present when solid elements are in the model.

To display the solid stresses on a displaced model, select the **Solid Stress Contour** tool in the **View Results** group on the **Results** ribbon tab in the **Postprocessing** workflow.

<table>
<thead>
<tr>
<th><strong>Property</strong></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Load Case</strong></td>
<td>Select the Load Case for which the stress is to be displayed.</td>
</tr>
<tr>
<td><strong>Stress Type</strong></td>
<td>Select the <strong>Stress Type</strong> from the drop down list. Once we select these, the <strong>Maximum</strong> and the <strong>Minimum</strong> values of that stress under the selected load case are displayed on the dialog box.</td>
</tr>
<tr>
<td><strong>Contour Type</strong></td>
<td>The <strong>Normal</strong> and <strong>Enhanced</strong> buttons indicate how the Stress Contour is drawn. The <strong>Normal</strong> contour option uses the stress points at each corner of the solid along with the center stress to calculate the contour. The <strong>Enhanced</strong> contour option uses the same points as the Normal contour plus the interpolated stress at the mid point of the edges. The second option takes more time to generate but is more accurate.</td>
</tr>
<tr>
<td><strong>Absolute Values</strong></td>
<td>causes the stress values to be compared based on the absolute values, rather than algebraic values. If this option is checked, stress values of +10 units and -10 units will be in the same range.</td>
</tr>
<tr>
<td><strong>Index Based on Center Stress</strong></td>
<td>the stress range of the index is based on the stress values obtained at the center of the elements.</td>
</tr>
<tr>
<td><strong>View Stress Index</strong></td>
<td>displays the legends of the colors with stress values at the side of the screen.</td>
</tr>
<tr>
<td><strong>Use Custom Limits</strong></td>
<td>By default, 15 contours are plotted with the maximum and minimum values determined by the program based on the results for the selected load case and stress type. Set the Use Custom Limits option to specify <strong>Minimum</strong> and <strong>Maximum</strong> values as well as <strong>No. of values</strong> to use for equal divisions.</td>
</tr>
</tbody>
</table>
Use Custom Divisions

By default, the values of the stresses used for color gradation are obtained by taking the Maximum and Minimum and dividing the difference equally into as many parts as is defined No. of values.

Setting the Use Custom Divisions allows you to directly type the cut-off values for each color.

Tip: Take care to ensure that they are in ascending order.

Re-Index stress range for new view

When individual elements are isolated from the full model then the stress distribution for the entire element may be displayed in a single color. This is because the variation in stresses across the element may not have been appreciable with respect to the entire model and the entire stress range may have been represented by a single color in the full model. This may create problem because the users in such cases will not be able to view the stress variation across the element. To take care of this The re-index stress range for new view option has been included which automatically regenerates the stress index for any newly created view which will allow the users to view the stress variation in much more detail.

Show Displaced Shape

Displays the solid stresses on a displaced model.

Related Links

- M. To display master nodes (on page 638)
- P. To display plate results contours (on page 2217)

Orientation dialog

Used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.

Opens when the Orientation tool is selected in the Rotation group on the View ribbon tab.

View group

Isometric — This option will display the structure in default isometric view (with 70 degree Elevation angle and 330 degree Rotation Angle). An isometric view uses effectively an infinite distance to the structure.
Perspective — This option will display the perspective view of the structure. The viewing distance and angles can be changed by changing one of the three options: Distance to Structure, Elevation Angle, or Rotation Angle.

Distance to Structure
This value indicates the distance of the eye (or camera) from the structure in the perspective view. You may type this value or use the spin control to increase or decrease the current value.

Tip: This distance should be greater than the extents of a volume containing the structure in order to prevent distorting the view.

Elevation Angle
This value indicates the rotational angle of the eye about an axis which is lying horizontal (left to right) in the plane of the screen and passing through the center of the screen. You may type this value or use the spin control to increase or decrease the current value.

Rotation Angle
This value indicates the rotational angle of the eye about an axis which is lying horizontal (left to right) in the plane of the screen and passing through the center of the screen. You may type this value or use the spin control to increase or decrease the current value.

Spin Angle
This value indicates the rotational angle of the eye about an axis which is perpendicular to the plane of the screen (in to out) and passing through the center of the screen. You may type this value or use the spin control to increase or decrease the current value.

Default Orientation To
The orientation of the structure may be set to a view defined by one of the following options:

- **Side** — View From + X (right)
- **Front** — View From + Z (front)
- **Plan** — View From + Y (top)
- **3D** — isometric angle (default)

Apply Immediately
Any change made through the Orientation dialog may be immediately reflected in the current view when this option is selected.

Use center of the structure as the rotational center
When this options is selected, STAAD.Pro will consider the center of the structure as the rotational center. The axis of rotation can be manually input by clearing this box and then providing the Rotational Center values.

Rotational Center
The axis of rotation can be manually input by checking this box and specifying the exact X, Y and Z coordinates in the Rotational Center boxes.

Tables dialog
Used to display and close different tables, such as Node coordinates, Beam incidences, Node displacements, etc. irrespective of the current page.

Opens when the Tables tool is selected in the Windows group on the View ribbon tab.
Tables  Check the associated boxes for the tables you want to display. The boxes are automatically checked for the currently displayed tables. To close a table, leave the associated box blank.

List  These check boxes control which tables are listed for selection from the **Tables** list. To display the list of available input related tables, set the **Analysis Input** option. To display the output related tables, set the **Analysis Results** option.

**Open View** dialog

Used to open a previously saved structural view window.

Opens when the **Open View** tool is selected in the **Views** group on the **View** ribbon tab.

- **Views list**  A list of saved views is displayed here. Select the View you want to open.
- **Do you want to:** options  Used to determine whether the selected view would be opened in a new window or whether it would replace the current view.
- **OK**  Opens the selected view, using the selected option.
- **Cancel**  Closes the Open View dialog without opening a view.
- **Help**  Opens the Help window.

**Options** dialog

Used to control display items such as fonts, colors and styles for different labels, units, etc. The tabs of the dialog are explained in the following pages.

Opens when **Display options** is selected on the **Settings** tab in the Backstage view.
**View Highlight tab**

Controls the simultaneous display of several view windows on screen. STAAD allows us to create multiple "child" view windows from the default "Whole Structure" ("parent") view. Typically, the "parent" view would display the entire structure, while the "child" windows would display selected portions of the structure in various orientations.

When one or more "child" windows are displayed on screen along with the "parent" window, the parent window highlights the portion of the structure contained in a "child" window, whenever that child window is active (gets focus). The next figure shows that the View 1 window is active and the portion of the structure contained in the View 1 window is highlighted in the Whole Structure window.

The View Highlight tab controls how the Whole Structure window highlights the portion of the structure contained in a "child" window. In the above figure, the Highlight View in Whole Structure Window group is related to saved views, while the Show Zoomed Area in Parent View group is related to the view window created using the Zoom Dynamic option.

- **Highlight**: When checked, the 'Parent' or Whole Structure window highlights the portion of the structure contained in the "child" window.
- **Width**: Controls the width of the highlighted portion of the structure. The associated color button allows selection of the highlight color.

**Tolerance tab**

**Tolerance**

Controls the minimum allowable spacing between two adjacent nodes so that the program does not generate errors of duplicate or overlapping joints. Type a Tolerance in the associated edit box.

**Split member if new node is added on the member**

Among the various methods available for adding a beam in the Modeling mode, two of the methods are using:

- **a.** The Beam sub-page of the Geometry page from the left side of the screen, and,
- **b.** Snap/Grid Node - Beam option of the Geometry menu from the top of the screen.

In these methods, if the start or end point of a member being drawn (let us call this member B) happens to fall within the span point of an existing member (let us call this member A), the setting chosen for this option will determine what happens to member A.

If the above option is switched on, the member A will be split into 2 members, and there will now be a total of 3 members incident from this common point.

If it is switched off, the member A will not be split, and the structure will be created as though the 2 members are NOT connected at the point in question.

One may test this by doing the following. Let us say that member A goes from (0,0,0) to (0,10,0), and that member B goes from (0,5,0) to (7,5,0).

If one were to use one of the methods (a) or (b) described above, it will be evident that leaving the option on will cause 3 members to be created, 5,5 and 6 units long. These 3 members will be connected at the common point (0,5,0).

Leaving the option off will cause only 2 members to be created, 10 and 7 units long respectively.
Here is another example of the relevance of this setting. Let us assume that a building is being assembled in two steps. First, the beam-column skeleton is created. Following that, the slabs and walls are created using the meshing capabilities of the Geometry | Structure Wizard option. If any of the nodes of the element mesh fall within the span lengths of the beams and columns, those beams and columns will either be split, or left unchanged, depending upon the settings here.

**Tolerance for Warped Plate Element Detection**

Used to specify a tolerance for the plate warping check. By default, plate warping is reported when there is a 30 degree angle (or larger) between the normal to two triangles which are formed by splitting a quad element. This splitting can be done two ways so there are four triangles. If any have more than the tolerance angle (in degrees) between them, then an error is reported. See “To check for warped plates” (on page 887) for how this value is used in the interface.

Set the **Save As Default** option to make this the default tolerance for warped plate checks.

**Tip:** Alternatively, warped plates can be checked by the STAAD engine using the `SET PLATE FLATNESS TOLERANCE` command. However, checking for warped plates in the User Interface allows you to quickly correct any plates prior to running an analysis.

**Tables tab**

Used to choose the font used in various tables.

- **Sample**  Displays the current typeface settings used for Tables.
- **Font…**  Opens the Font dialog (on page 2941), which is used to select font type, size, style, etc. used in tables.

**Assign Dlgs tab**

Used to control the display of labels for Property, Load, Support etc. while these are being assigned.

- **Show all labels when dialog is open**  Displays labels for all assigned properties, loads, etc. when the corresponding dialog is opened.
- **Only show labels for selected items in dialog**  Displays labels for only the selected items.
- **Show all labels, with labels for selected items in the following color**  Displays labels for the selected items in the specified color. Displays all other labels in default color.

**Beam Labels/Plate Labels/Solid Labels/Load Labels/Load Icons/Annotation/Dimension tabs**

These tabs offer options for labeling beams, plates, solids, etc. The **Beam Labels** tab controls how the labels for the beams (including the dimension of the members) appear on the screen, and is explained below.

- **Style**  Select any of the pre-defined styles for displaying the beam numbers.
- **Separator, Reference**  These options control how the property number reference is displayed with the member numbers.
- **Horizontal Alignment, Vertical Alignment**  These items determine the relative position of the label with respect to the member.
**Font**
Opens the *Font dialog* (on page 2941), which is used to select font type, size, style etc. for the label.

**Opaque**
When checked, the labels behave as opaque objects, i.e. everything behind them become hidden.

**Angle Text**
When checked, displays the labels inclined along the members.

**Structure Units/Section Units/Force Units tabs**
The *Structure Units*, *Section Units* and *Force Units* tabs offer options for specifying units for the structure, sectional views, forces, etc. The *Section Units* tab is described below and is shown in the next figure.

- **Dimension**
Used to set the length units in which dimension lines are drawn on the screen. This feature dictates the values that will be displayed for the *Dimension Beams* facility as well as the *Display Node to Node Dimension* facility of the *Tools* menu.

**Note:** Changing units on these tabs does not change the input units in the Input Command File, the units are changed in the graphical display only. To change units in the Input Command File, see *Tools | Set Current Input Unit...* menu.

The *Section Units* tab allows you to specify the units to display different data items such as sectional Area, Moment of Inertia, etc. Select the appropriate units from the available drop down list and specify the number of digits to be displayed after the decimal point using the *Show* spin boxes.

- **Dimension**
Used to set the length units in which certain property values are displayed on the screen. If you choose the *General* page followed by the *Property* page along the left side of the screen, a button called *Values* will be present along the right side of the screen. If properties have already been assigned to any members, you can examine the property values by clicking on the *Values* button. The terms $D$, $Bf$, $Tf$ and $Tw$ use the length settings of this facility. The same table may also be displayed by going to *View | Tables | Section Properties*.

- **OK**
Applies all changes made and closes the dialog

- **Cancel**
Closes the dialog without applying any changes.

- **Apply**
Applies all changes made. Only active if changes have been made since the dialog was opened or the Apply button was last clicked.

- **Help**
Opens the STAAD.Pro help window.

**Font**
Used to control the text display style.

Opens when the *Font* button is clicked in various dialogs.
Font: Sets the typeface. Select from the scroll list or simply type in the field to search for an installed font name.

Font style: Select font styles or combination of styles from the list.

Size: Sets the text size.

Effects: (Not available for all Font dialogs) Add Strikeout and Underline effects to the text.

Color: (Not available for all Font dialogs) Select a standard color to use for the font.

Script: Select the script set used.

OK: Applies the font changes and closes the dialog.

Cancel: Closes the dialog without applying any changes.

**Color Manager** dialog

Used to specify colors of different items such as analysis results.

Opens when either:

- **Set Structure Colors** is selected on the **Settings** tab in the Backstage view, or
- the **Set Structure Colors** tool is selected in the **Options** group on the **View** ribbon tab.
**View Highlights**  
Opens a color selection dialog, which is used to specify a highlighting color for selected objects.

**Entities**  
Opens the **Define Colors** dialog, which is used to color code structural entities by specific attributes.

---

Specify the type of entity (Beam, Plate or Solid) whose colors are to be changed under the Select **Structural Entity** list box. Under the **Color By** option, select the type of attribute for which the colors will be arranged according to. To change the color of a particular attribute (for example, if Beam Properties option is selected, all the properties for that model will be listed), double-click on the color to bring up the color palette dialog box. Select a new color for that particular attribute.
Note: The colors of the entities can also be changed through the Structure tab of the Diagrams dialog.

**Forces**
Opens a color selection dialog, which is used to specify a highlighting color for forces shown in analysis results.

**Limits**
Opens a color selection dialog, which is used to specify a highlighting color for limits in analysis results.

**Results**
Opens a color selection dialog, which is used to specify a highlighting color for results.

**Node Dimension**
Opens a color selection dialog, which is used to specify a highlighting color for node-to-node dimensions.

**Current Structure Color Scheme**
The following options are available for applying the selected color changes:

- Structure Only
- Force Limits
- Design Results

**Tool Tip Options** dialog

Used to display any customized input or output information about a node, beam, plate or solid element by simply placing the mouse over the structural entity.

The structural tool tips include the maximum and minimum nodal displacements and beam end forces out of all the load cases (envelope of displacements or forces). You can now place the mouse over a node or beam and have a tool tip report these values. Tool tips are similar to the information box displayed when the mouse hovers over a toolbar icon.

Opens when **Structural Tool Tip Options** is selected on the **Settings** page of the Backstage view.

**Note:** The tool tips automatically display the results for the active load case. All values are reported in the current display units.

**Show Tool Tip**
Turns the structural tool tips on or off.

**Tip Delay**
signifies the amount of time it takes from when the mouse hovers over an entity to when the tool tip actually pops up. This number is expressed in milliseconds (i.e., 1000 = 1 second).

**Tool**
Select the Node or Beam item from the left hand side. On the right hand side, there will be new options to enable the display of the maximum and minimum nodal displacements or beam end forces out of all the load cases (i.e., an envelope).

**Options**
The options (items that can be displayed) for each entity are shown here. A check mark signifies that the particular data item will be displayed in the tool tip.

**Note:** An option with a “+” next to it signifies that further options can be enabled or disabled.

**Note:** A red “X” indicates the data will not be shown in the tool tip. Simply click on the check box to turn an option on or off.
The resulting tool tip for nodes with the Node Number, Displacement, and Support options set.

![Tool tip example](image)

*Figure 326: Example tool tip at a support*

**Select tab**

Contains tools used for selecting objects in your analytical model.

**Table 295: Cursors group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes Cursor</td>
<td>Used to graphically select nodes.</td>
</tr>
<tr>
<td>Beam Cursor</td>
<td>Used to graphically select beams.</td>
</tr>
</tbody>
</table>

**STAAD.Pro User Manual**
<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate Cursor</td>
<td>Used to graphically select plates.</td>
</tr>
<tr>
<td>Solid Cursor</td>
<td>Used to graphically select solids.</td>
</tr>
</tbody>
</table>
| Geometry Cursor         | Used to graphically select nodes, members and elements of the structure simultaneously.  
                          | To select nodes, members, or elements using the Geometry Cursor, simply click on the desired structural components. To select multiple nodes, members and elements, hold <Ctrl> while selecting. We may also select the structural components graphically by creating a window on screen with the cursor around these components. |
| Members Cursor          | Used in Steel Design or Concrete Design to graphically select all those beams defined as a same member in the member set up of member design, simultaneously.  
                          | Members may be user defined or may be generated automatically using the Auto Form Member tool.  
                          | To select all the beams defined as a same member using the Member Cursor, just click on one beam. The other beams having the same member name as the selected one will automatically be selected. To select multiple physical members, hold down <Ctrl> while selecting. You may also select the physical members graphically by dragging a fence area around these physical members using the cursor. |
| Plates & Solids Cursor  | Used to graphically select both plate and solid elements.                  |
| (Select) Text           | Used to add comments and titles to pictures and result diagrams. The added text can be plotted, too.  
<pre><code>                      | The inserted text can be deleted, moved, and modified using the text cursor. Refer to the Insert Text on the Utilities ribbon for a detailed description on inserting text and modifying it using the text cursor. |
</code></pre>
<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Previous</td>
<td>Selects the last objects selected if they have been deselected or if the selection set has changed.</td>
</tr>
<tr>
<td>Load</td>
<td>Used to modify any load already applied on the model by double clicking it. When selected, the mouse pointer changes to the Load Edit Cursor.</td>
</tr>
</tbody>
</table>

**Table 296: Geometry group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>All</td>
<td>Selects all nodes, members, elements, and solids in the model.</td>
</tr>
<tr>
<td>Inverse</td>
<td>Used to select all objects but the ones which are currently selected.</td>
</tr>
<tr>
<td>List</td>
<td>Opens the Select Geometry dialog, which is used to select one or more model entities (other than nodes) from a list of all entities in the model.</td>
</tr>
<tr>
<td>Parallel</td>
<td>Used to select all beams, elements, and surfaces which are parallel to a specified global axis. Select the desired global axis from the sub-menu.</td>
</tr>
</tbody>
</table>
  * To XY
  * To YZ
  * To XZ |
### Ribbon Control Reference

**Select tab**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Connected" /> <strong>Connected</strong></td>
<td>Used to select all the entities that are connected to (i.e., have a common node with) any particular node, beam, plate, or solid. A dialog opens prompting you to select the node, beam, plate, or solid number. You can select to apply immediately or click <strong>OK</strong> to see all the model entities that are connected to the selection.</td>
</tr>
<tr>
<td><strong>Tip:</strong> Options for geometry objects not contained in the current mode are disabled.</td>
<td></td>
</tr>
<tr>
<td><img src="image2.png" alt="Highlight" /> <strong>Highlight</strong></td>
<td>Opens the <strong>Visual Check</strong> dialog, which is used to sequentially highlight a specific group of entities (beams, plates, or solids) in numerical order. Controls on the speed of the selection are also included.</td>
</tr>
</tbody>
</table>

### Table 297: Nodes group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image3.png" alt="All" /> <strong>All</strong></td>
<td>Selects all nodes in the model.</td>
</tr>
<tr>
<td><img src="image4.png" alt="Inverse" /> <strong>Inverse</strong></td>
<td>All nodes in the selection set are deselected and any previously unselected nodes are added to the selection set.</td>
</tr>
<tr>
<td><img src="image5.png" alt="List" /> <strong>List</strong></td>
<td>Opens the <strong>Select Nodes</strong> dialog, which is used to select one or more nodes from a list of all nodes in the model. A free list of node numbers may also be specified.</td>
</tr>
<tr>
<td><img src="image6.png" alt="Supports" /> <strong>Supports</strong></td>
<td>Selects all supported nodes in the model.</td>
</tr>
</tbody>
</table>
### Table 298: Beams group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>All</td>
<td>Used to select all beams.</td>
</tr>
<tr>
<td>Inverse</td>
<td>All beams in the selection set are deselected and any previously unselected beams are added to the selection set.</td>
</tr>
<tr>
<td>List</td>
<td>Opens the <strong>Select Beams</strong> dialog, which is used to select one or more beams from a list of all beams in the model.</td>
</tr>
</tbody>
</table>
| Parallel  | Select beams that are parallel to the selected global axis.  
- To XY  
- To YZ  
- To XZ |
| Connected | Select beams that connect to the selected object type.  
- To Node  
- To Beam  
- To Plate  
- To Solid |

**Tip:** Options for geometry objects not contained in the current mode are disabled.

### Table 299: Plates group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>All</td>
<td>Selects all plates in the model.</td>
</tr>
</tbody>
</table>
### Plate Selection Tools

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inverse</td>
<td>All plates in the selection set are deselected and any previously unselected plates are added to the selection set.</td>
</tr>
<tr>
<td>List</td>
<td>Opens the Select Plates dialog, which is used to select one or more plates from a list of all plates in the model.</td>
</tr>
</tbody>
</table>
| Parallel  | Select plates that are parallel to the selected global axes plane.  
• To XY  
• To YZ  
• To XZ |
| Connected | Select plates that connect to the selected object type.  
• To Node  
• To Beam  
• To Plate  
• To Solid |

**Tip:** Options for geometry objects not contained in the current mode are disabled.

### Solid Selection Tools

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>All</td>
<td>Selects all solids in the model.</td>
</tr>
<tr>
<td>Inverse</td>
<td>All solids in the selection set are deselected and any previously unselected solids are added to the selection set.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
</tr>
<tr>
<td>List</td>
<td>Opens the <strong>Select Solids</strong> dialog, which is used to select one or more solids from a list of all solids in the model.</td>
</tr>
<tr>
<td>Connected</td>
<td>Select solids that connect to the selected object type.</td>
</tr>
<tr>
<td></td>
<td>• <strong>To Node</strong></td>
</tr>
<tr>
<td></td>
<td>• <strong>To Beam</strong></td>
</tr>
<tr>
<td></td>
<td>• <strong>To Plate</strong></td>
</tr>
<tr>
<td></td>
<td>• <strong>To Solid</strong></td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> Options for geometry objects not contained in the current mode are disabled.</td>
</tr>
</tbody>
</table>

**Table 301: Filter group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter</td>
<td>Used to select multiple types of geometric entities (nodes, beams, surfaces, etc.) with specific attributes in one pass. This will reduce the time required to create new views and help quickly identify the location of certain entities on your structure.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Before this cursor can be used, the actual filter parameters must be defined in advance in the <strong>Selection Filters</strong> dialog.</td>
</tr>
</tbody>
</table>

**Table 302: Attributes group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group</td>
<td>Opens the <strong>Select Groups</strong> dialog, which is used to select objects in a named group.</td>
</tr>
<tr>
<td>Property Name</td>
<td>Used to select nodes and members based on specifications associated with them. A number of specification types are included in the sub-menu list.</td>
</tr>
</tbody>
</table>
### Tool name | Description
--- | ---
Missing Property | Used to select beams, plates, or solids that lack critical input data (Property, Density, Elasticity, Poisson’s ratio, and Alpha) in the active view.

#### Table 303: Modes group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag Box</td>
<td>Click to activate the drag box selection mode.</td>
</tr>
<tr>
<td>Drag Line</td>
<td>Click to activate the drag line selection mode.</td>
</tr>
<tr>
<td>Region</td>
<td>Click to activate the region selection mode.</td>
</tr>
</tbody>
</table>

### Visual Check dialog

Used to easily identify the location of entities in a certain section of the model. Many times, commands like assigning properties or releases are used for a certain set or range of entities which may be difficult to graphically observe because of their close proximity to each other. This feature may help alleviate difficulties in checking the properties of a concentrated group of entities.

Commands in the dialog to control the speed of the automatic selection process are described below.

Opens when the **Geometry > Highlight Entities Sequentially** tool is selected in the Selection group on the Geometry ribbon tab.

- **Start With list**
  - Select the entity type to highlight.
- **Currently Selected**
  - Displays the currently selected entity
  - Automatically selects the type of entity chosen in the Start With list in a sequential order (by entity number).
Pauses the sequential selection operation. Inactive when the automatic selection is not active.

Increases the speed at which the automatic selection occurs. The selection process becomes faster with each press of this button.

Decreases the speed at which the automatic selection occurs. Each time this button is pressed, the selection process becomes slower.

Increments the selection manually one entity at a time. If this is not highlighted, the Pause button must be pressed to stop the automatic selection process.

Decrements the selection process manually one entity at a time. If this is not highlighted, the Pause button must be pressed to stop the automatic selection process.

Stop

Stops the selection process.

Query

Opens the entity query dialog for the currently selected entity.

Exit

Closes the Visual Check dialog.

**Select Nodes** dialog

Used to select one or more nodes from a list of all nodes in the model. A free list of node numbers may also be specified.

**list of nodes**

Select the desired nodes by clicking on their number in the displayed list. To select sequential nodes, either click and hold the mouse button while dragging over the numbers or hold down <Ctrl> while clicking.

**Selection Type**

To select nodes by numbers in the list, select the **Select from list by cursor** option. To select nodes with a typed list of node numbers, click the **Select using typed list** option then click **Select Listed Entities**.

**Select Listed Entities**

Click to select nodes listed in the **Enter list** box.

**Enter list**

Specify node numbers for selection. You may use to to represent a range of sequential node numbers. Separate values by either spaces or commas.

**Tip:** Clicking in this box selects the Select using types list option for Selection Type automatically.

**Note:** Similar features exist for selecting by lists of beams, plates, solids, surfaces, and geometry.

Opens when the **Nodes > Node List** tool is selected in the **Selection** group on the **Geometry** ribbon tab.

**Select Groups** dialog

Used to select members in a named group.

Opens when the **Group Selection** tool is selected in the **Selection** group on the **Selection** ribbon tab.
**Group List**  Select a single group to highlight the component model elements in the View window. Multiple groups may be selected by pressing <Ctrl>.

**Show Label** If labels pertaining to certain entities are switched on (e.g., beam labels), the labels will be displayed only for members with a **Show Label** check box set. Clear the check box for any group for which you do not want labels displayed.

**Close** Closes the dialog.

---

**Specification tab**

**Table 304: Beam Profiles group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Database" /></td>
<td>Click the drop-down arrow to display a gallery list of all database section tables. Click a country name or material to open the corresponding section table dialog.</td>
</tr>
<tr>
<td><img src="image" alt="Prismatic" /></td>
<td>Opens the <strong>Property</strong> dialog, which is used to assign Circular, Rectangular, Tee, Trapezoidal, General(arbitrary), etc. cross sections to the members.</td>
</tr>
<tr>
<td><img src="image" alt="Tapered" /></td>
<td>Opens the Tapered I Property dialog, which is used to specify an I-section having a varying depth over the length of the member.</td>
</tr>
</tbody>
</table>
## Tool name | Description
--- | ---
User Table Manager | Opens the User Table Manager dialog, which is used for adding and managing previously created user table sections to the structure.
Assign from User Table | Opens the User Property Table dialog, which is used to assign previously created user table sections to the structure.
**Note:** If you have not created any user-provided tables, then you will be asked if you would like to create one first.
Assign Profile | Opens the Assign Profile dialog, which is used to select a section profile and optional material to a member for member selection during design.

### Table 305: Plate Profiles group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
Plate Thickness | Opens the Plate Element/Surface Property dialog, which is used to provide plate element properties (thickness) with or without the material specification. Plates can have a different thickness at each node. Surfaces have a constant thickness.

### Table 306: Material Constants group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
Young's Modulus | Opens the Material Constant - Elasticity dialog, which is used to assign the modulus of elasticity, $E$, based on a predefined material value or a custom value. |
Shear Modulus | Opens the Shear Modulus - $G$ dialog, which is used to assign the shear modulus, $G$, based on a predefined material value or a custom value. |
Poisson's Ratio | Opens the Material Constant - Poisson's Ratio dialog, which is used to assign Poisson's ratio, $\nu$, based on a predefined material value or a custom value. |
Density | Opens the Material Constant - Density dialog, which is used to assign the material density, $\gamma$, based on a predefined material value or a custom value. |
Thermal Coefficient | Opens the Material Constant - Alpha dialog, which is used to assign the thermal coefficient, $\alpha$, based on a predefined material value or a custom value. |
<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Damping Ratio</strong></td>
<td>Opens the Material Constant - Damping Ratio dialog, which is used to assign the damping ratio, c, based on a predefined material value or a custom value.</td>
</tr>
</tbody>
</table>

**Table 307: Specifications group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Node &gt;</strong></td>
<td></td>
</tr>
<tr>
<td>Add Master/Slave Specification</td>
<td>Opens the Node Specification dialog to the Master/Slave tab, which is used to select a master node and the directions which other nodes follow actions.</td>
</tr>
<tr>
<td>Delete Master/Slave Specification</td>
<td>Used to delete a master/slave specification.</td>
</tr>
<tr>
<td><strong>Beam &gt;</strong></td>
<td></td>
</tr>
<tr>
<td>Beta Angle</td>
<td>Opens the Beta Angle dialog, which is used to specify the beta angle for members.</td>
</tr>
<tr>
<td>Beam Reference Point</td>
<td>Opens the Reference Point dialog, which is used to specify a reference point for the members from which the program calculates the beta angle.</td>
</tr>
<tr>
<td>Cable</td>
<td>Opens the Member Specification dialog to the Cable tab, which is used to assign standard cable and nonlinear cable specifications.</td>
</tr>
<tr>
<td>Compression Only</td>
<td>Opens the Member Specification dialog to the Compression tab, which is used to define compression-only members. These members are capable of carrying compressive forces only.</td>
</tr>
<tr>
<td>Tension Only</td>
<td>Opens the Member Specification dialog to the Tension tab, which is used to define tension-only members. These members are capable of carrying tensile forces only.</td>
</tr>
<tr>
<td>Truss</td>
<td>Opens the Member Specification dialog to the Truss tab, which is used to specify truss members. These members are capable of carrying axial forces only.</td>
</tr>
<tr>
<td>Imperfection</td>
<td>Opens the Member Specification dialog to the Imperfection tab, which is used to assign camber or drift values to members.</td>
</tr>
<tr>
<td>Cracked Property</td>
<td>Opens the Member Specification dialog to the Property Reduction Factors tab, which is used to assign reduced (i.e., “cracked”) section property factors to members.</td>
</tr>
<tr>
<td>Release</td>
<td>Opens the Member Specification dialog to the Release tab, which is used to assign releases, partial moment releases, or spring values to member ends.</td>
</tr>
</tbody>
</table>
### Tool name | Description
---|---
Offset | Opens the **Member Specification** dialog to the **Offset** tab, which is used to assign member end offset values to a member end.
Inactive | Opens the **Member Specification** dialog to the **Inactive** tab, which is used to instruct the analysis and design engine to ignore a member for analysis.
Fireproofing | Opens the **Member Specification** dialog to the **Fire Proofing** tab, which is used to assign block or contour fire proofing material to a member for weight calculations.
Plate Reference Point | Opens the **Plate Reference Point** dialog, which is used to specify options for determining the general direction of the local Z axis of elements,
Release | Opens the **Plate Specs** dialog to the **Release** tab, which is used to assign released degrees of freedom at a plate node.
Ignore Inplane Rotation | Opens the **Plate Specs** dialog to the **Ignore Inplane Rotation** tab, which is used to override the value that STAAD determines for the stiffness associated with the local axis Mz degree of freedom for plate elements.
Plane Stress | Opens the **Plate Specs** dialog to the **Plane Stress** tab, which is used to model plate elements as Plane Stress elements.
Ignore Stiffness | Opens the **Plate Specs** dialog to the **Ignore Stiffness** tab, which is used to instruct the program to neglect the stiffness of the selected plates while assembling the stiffness matrix. Hence, the stiffness contribution of the selected plates is not considered. This is similar to the Inactive command for the members.

### Table 308: Supports group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fixed</td>
<td>Opens the <strong>Create Support</strong> dialog <strong>Fixed</strong> tab, which is used to create a fixed support tag and (optionally) to assign that to selected nodes. A Fixed support is restrained in all 6 degrees of freedom.</td>
</tr>
<tr>
<td>Pinned</td>
<td>Opens the <strong>Create Support</strong> dialog <strong>Pinned</strong> tab, which is used to create a pinned support tag and (optionally) to assign it to selected nodes. A Pinned support is restrained in all three translational degrees of freedom and free in the 3 rotational degrees of freedom.</td>
</tr>
</tbody>
</table>
### Tool name | Description
---|---
Custom | Opens the **Create Support** dialog **Fixed But** tab, which is used to create various types of roller, hinge and spring supports with specified restrained degrees of freedom and to assign them to selected nodes.

Foundation | Opens the **Create Support** dialog **Foundations** tab, which is used to create spring supports for independent footings and mat foundations and to assign them to selected nodes.

One-way Spring | Opens the **Create Support** dialog **Tension/Compression Only Spring** tab, which is used to designate certain support springs as Tension-only or Compression-only springs.

Other Supports > | Inclined | Opens the **Create Support** dialog **Inclined** tab, which is used to create supports that offer restraints in an axis system that is inclined with respect to the global axis system.

| Multilinear Spring | Opens the **Create Support** dialog **Multilinear Spring** tab, which is used to enter displacement and spring stiffness data pairs for a multilinear spring definition, which is then used for a Fixed But support type.

Enforced | Opens the **Create Support** dialog **Enforced** tab, which is used to create a fixed-condition support defined in terms of being stiff springs.

Custom Enforced | Opens the **Create Support** dialog **Enforced But** tab, which is used to create a fixed-condition support defined in terms of being stiff springs with the option to release selected degrees of freedom.

**Table 309: Tools group**

| Tool name | Description |
---|---|
| | Launches the **Section Wizard** application. |
### Specifications - Whole Structure dialog

Used for defining specifications and assigning them to nodes, members and elements.

Opens when either:

- the Specifications page is selected, or
- the Specifications Layout tool in the Specifications group on the Specifications ribbon tab is selected

**Specification list**

All Specifications that have been defined for the model are listed here.

**Note:** Physical Member specifications are designated with either PMEMBER or Physical tags.

**Highlight Assigned Geometry**

When this box is checked and you click on a specification in the Specifications dialog box, members that have been assigned the specification will appear highlighted in the structure window. For example in the previous figure, if you click on the MEMBER TRUSS specification, all truss members will be highlighted.

**Edit/Delete**

Used to modify a previously assigned attribute, or delete it altogether.

**Node…**

Opens the Node Specification dialog (on page 2960), which is used to specify rigid links or specialized linkages in the structure.

**Beam…**

Opens the Member Specification dialog (on page 2962), which is used for specifying frame member specifications.

**Plate…**

Opens the Plate Specs dialog (on page 2967), which is used to specify finite element specifications.

**Toggle Specification**

This is a switch setting which turns on what is called the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens.

- Click on the member or element once - the attribute gets assigned.
- Click on the same member or element a second time - the attribute gets de-assigned.
- Click on the same member or element again, - the attribute gets re-assigned.

Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.

**Assignment Method**

The options under the Assignment Method are used to assign specifications to joints, members and elements. The assignment method can be one of Assign To Selected Beams, Assign To View, Use Cursor To Assign or Assign To Edit List.
- **Assign To Selected Beams** - To assign a specification to selected nodes, members or elements, first select the specification from the **Specifications** dialog box. The specification selected is highlighted. Then select the nodes, members or elements to which this specification is to be assigned. This is done by going to the **Select** menu, then choosing the appropriate cursor option. Select the desired joints, members or elements using the cursor. When all desired nodes, members or elements are selected, click the **Assign To Selected Beams** radio button, then click the **Assign** button. Note that the label for this button changes depending on whether nodes, members or elements are selected.

- **Assign To View** - To assign a specification to all nodes, members or elements in a view, first select the specification from the **Specifications** dialog box. The selected specification is highlighted. Select the **Assign To View** radio button, then click the **Assign** button. All nodes, members or elements in the structure are assigned this specification.

- **Use Cursor To Assign** - To assign a specification to nodes, members or elements using the cursor, first select the specification from the **Specifications** dialog box. The selected specification is highlighted. Select the **Use Cursor To Assign** radio button, then click the **Assign** button. The button will appear depressed and the label will change to **Assigning**. Use the appropriate cursor (from the Select menu) to assign the selected specification to the individual nodes, members or elements. Click on the **Assign** button again to finish.

- **Assign To Edit List** - To assign a specification using a typed list of node, member or element numbers, first select the specification from the **Specifications** dialog box. The selected specification is highlighted. Select the **Assign To Edit List** radio button, then type the list of node, member or element numbers and click the **Assign** button.

**Related Links**
- [M. To add a member specification](on page 797)

**Node Specification** dialog

Used to specify rigid links or specialized linkages in the structure. This facility can be used to model special structural elements such as ties or a floor diaphragm which makes the floor rigid for in-plane movements. The process is to select a node to act as the master and then degrees of freedom for which the slave nodes are linked to the master. Internally, STAAD will perform an automatic bandwidth reduction.

**Note:** Refer to [TR.28 Rigid Diaphragm Modeling](on page 2525) for additional details.

Opens when **Node** is clicked on the **Specifications - Whole Structure** dialog.

**Note:** STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) and higher is capable of defining a rigid floor diaphragm without the need to specify a Master Node. This feature is only available by manually editing the STAAD input file. Refer to [TR.28.2 Floor Diaphragm](on page 2526) for additional details.

**Master/Slave tab**

**Master Node** It is called as such because the displacements of the slaved node will be based on the displacement of this Master node. From the drop-down list, select the node which should serve as the master node.
Ribbon Control Reference
Specification tab

Slaved Directions

- **Rigid** - This is a means of defining that the link which connects the master with the slave has infinite stiffness for all the six degrees of freedom. In other words, if one were to consider an imaginary object that has infinite stiffness axially, for shear, torsion and bending, that would be the type of link which connects the master and the slave.

  **Note:** By default, STAAD.Pro keeps this box checked. Un-check this box to activate the options described below.

- **XY, YZ, and ZX** - This is a way of specifying a type of connection where the rigidity is limited to in-plane directions.

  The plane of this link is defined in terms of the global planes XY, YZ, or ZX. For example, if we have a concrete slab in a building, and the slab can be assumed to be nearly inflexible for translations along global X and global Z, and the rotation about global Y, that type of link would be called a "diaphragm" in global XZ.

  XY is equivalent to a master-slave relationship for the degrees of FX, FY and MZ. Similarly, YZ is equivalent to linking FY, FZ and MX, and, ZX is equivalent to linking FX, FZ and MY.

- **FX, FY, FZ, MX, MY, and MZ** - This provides a way for linking just some specific degrees of freedom the user desires. For example, if one wishes to link FY and MZ, check these two options.

---

**Floor Diaphragm tab**

**Diaphragm group**

- **No**
- **Type**
  - Select the diaphragm type from the drop-down list:
    - **Rigid** - a diaphragm that translates & rotates as a rigid body.

**Floor Level group**

Select one of the options as the method to define the floor level of the diaphragm.

- **Height**
  - Type the global coordinate value (elevation), in Y direction, to specify floor level.

- **YRange**
  - Select this option to define the floor level by upper and lower bounds of Y coordinates.

**YRange group**

Used when the **YRange** option is selected.

- **Minimum**
  - The lower bound of Y coordinates for the Y range.

- **Maximum**
  - The upper bound of Y coordinates for the Y range.

**Master Node Information group**

Select the method used to define the master node of the diaphragm.

- **Calculate**
  - The diaphragm center of mass is determined and a node is created at that location to use as the master node.

- **Select Master Node**
  - Select the existing node to use from the drop-down list.
Define Floor Range  Set this option to define the floor range as a XZ range of coordinates or by manually selecting coordinates.

Related Links
- M. To assign nodes to a floor diaphragm (on page 811)
- G.14 Rigid Diaphragms (on page 2336)
- M. To assign a rigid link between nodes (on page 810)

Member Specification dialog

Used to set and modify member (beam) specifications.

Opens when Beam is clicked on the Specifications - Whole Structure dialog.

This dialog contains multiple tabs, each of which is used to add a different member specification.

Release tab

Used to specify the end conditions of members by releasing specified degrees of freedom. Unless a release is specified, all members are rigidly connected the nodes (i.e., all degrees of freedom are restrained) except if the end is a cantilever end or supported.

Note: Refer to TR.22.1 Member Release Specification (on page 2488) for additional information.

Location
Specify either Start or End as the released joint Member Incidence of the member.

Release Type
Select either the Partial Moment Release or the Release option.

Partial Moment Release
Type a release Factor to define a partial release condition. The MP option is a means of specifying the same partial release for all the 3 moment degrees of freedom (MX, MY and MZ).

Alternately, use the MPX, MPY or MPZ options to apply different release factors in each direction.

Release
Click the Release radio button and check the boxes for the FX, FY, FZ, MX, MY or MZ directions to define the member release condition. You may also specify a Spring Constant along the six degrees of freedom by using the KFX, KFY, ... KMZ edit boxes. For example, to define a Spring Constant in the local X direction, enter the Spring Constant in the KFX edit box.

Tension tab

Used to define tension-only members. These members are capable of carrying tensile forces only.

There are no additional parameters for this member specification.

Property Reduction Factors tab

Used to specify a set of reduction factors to be applied on the calculated section properties such as area, moments of inertia, and torsional constant.
Ribbon Control Reference
Specification tab

Note: Refer to TR.20.10 Member Property Reduction Factors (on page 2485) for additional information.

Select the method of how to specify member property reduction factors:

- **Global** – specify the **Reduction Factors** to be used below, which will apply to any member regardless of material.
- **Code Specific** – select the building code to use reduction factors specific to load cases per that code which are only applied to concrete members:

  **IS1893 2016**

  **Reduction Factors**

  Provide a set of reduction factors between which will be used in the stiffness analysis. These factors are directly multiplied to the following section properties:

  - Reduction Factor for Cross sectional Area (RAX)

  **Note:** For IS1893 2016 reduction factors, the RAX input field is inactive as this property reduction is not mandated by that code for concrete members.

  - Reduction Factor for Torsion Constant (RIX)
  - Reduction Factor for Moment of Inertia, major axis (RIY)
  - Reduction Factor for Moment of Inertia, minor axis (RIZ)

  **Notes:** Reduction factors are considered for analysis only but not for design.

  Results using the reduced section properties are not available when using the member query feature.

  **Note:** Automated stiffness reduction analysis is not supported by the basic solver. STAAD.Pro Advanced is required for this feature.

Cable tab

Used to define cable members. Select either the Tension or Length option to specify a member tension. Tension-only springs are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them. Compression-only springs are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them.

Note: Refer to TR.23.2 Member Cable Specification (on page 2493) for more information.

**Initial TENSION**

Use this option to specify the initial tension in the cable as a force.

- **Average**
  The average tension in the cable is used (considered in advanced cable analysis only).

- **Start node**
  Tension is measured from the start node of the member (considered in advanced cable analysis only).

- **End node**
  Tension is measured from the end node of the member (considered in advanced cable analysis only).
Unstressed LENGTH
Use this option to specify the initial tension in the cable as a length for nonlinear cable analysis.

Factor in global X
Fwx
Multiplying factors on self weight components applied in the global X direction.

Factor in global Y
Fwy
Multiplying factors on self weight components applied in the global Y direction.
Tip: When Y is up, use a negative value to act as gravity.

Factor in global Z
Fwz
Multiplying factors on self weight components applied in the global Z direction.
Tip: When Z is up, use a negative value to act as gravity.

Truss tab

Used to specify truss members. These members are capable of carrying axial forces only.
There are no additional parameters for this member specification.

Compression tab

Used to define compression-only members. These members are capable of carrying compressive forces only.
There are no additional parameters for this member specification.

Offset tab

Used to rigidly offset a frame member end from a joint to model the offset conditions existing at the ends of frame members.

The actual beams and columns of a physical structure are represented by lines in the mathematical (computer) model. In the actual structure, a beam spans a distance which in the clear span between the faces of columns. But in the model, the line for the beam spans between the centerlines of the column. The half-depth portion of either column is considerably stiffer than the beam itself from the standpoint of bending. To take advantage of this additional stiffness, you may specify that the start and end faces of the beam are offset from the node by a distance equal to the half-column-depths.

Member offsets can be specified in other situations too. For example, consider:

- when a bracing member does not meet the node which is defined in its incidence list
- a girder and top slab in a bridge where the centerline of the girder is several inches below the centerline of the slab.

This facility is useful when you want to design the structural parameters of a member by considering the clear distance of the member between the supports (e.g., in the case shear forces or bending moments).

Refer to TR.25 Member Offset Specification (on page 2499) for additional information on member offsets.

Location
Specify either Start or End as the joint at which to specify an offset.
Direction

Select either the **Local** or **Global** axis system for assigning offset distances.

Offsets

Enter the offset distance from the joint in the three axis directions.

**Inactive tab**

Used to temporarily inactivate members for a specific analysis cycle. Inactivated members may be re-activated later for further processing.

Information on using this is available in [EX. US-4 Inactive Members in a Braced Frame](#) (on page 4390) or [EX. UK-4 Inactive Members in a Braced Frame](#) (on page 4670). This option is also explained in [TR.18 Inactive/Delete Specification](#) (on page 2445).

There are no additional parameters for this member specification.

**Fire Proofing tab**

Used to automatically consider the weight of fire proofing material applied to structural steel.

Refer to [TR.20.9 Applying Fireproofing on members](#) (on page 2482) for additional information on fireproofing.

**Fire Proofing Type**

Two types of fireproofing configurations are currently supported:

- **BFP (Block Fireproofing):**
  
  The fire-protection material forms a rectangular block around the steel section. The thickness specified is the minimum thickness which defines the outer block dimensions.

  ![Figure 327: BFP: Block Fireproofing](#)

- **CFP (Contour Fireproofing):**
  
  The fire-protection material forms a coating around the steel section. The thickness specified is a constant thickness around the section profile.
Figure 328: CFP Contour Fireproofing

**Thickness**  
Thickness (dimension in the figures above) in length units

**Density**  
Density of fireproofing material in (force / length$^3$) units

**Imperfection tab**

Used to define camber and drift specifications for selected members.

**Note:** Refer to TR.26.6 Member Imperfection Information (on page 2511) for additional details.

**Imperfection type**  
Select if whether the imperfection is either **Camber** or **Drift**. Drift is usually for columns and camber for beams.

**Local Direction**  
Specify if the camber or drift acts in either the member local **Y** or **Z** direction.

**Value**  
Specify the value for the camber or drift, defined as a ratio of member length/offset. Default ratio value is 300.

**Respect**  
(Camber option only) A non dimensional constant used to skip the camber imperfection calculation if the compressive load is small or EI is great or length is short. A combination of these terms is calculated and called EPSILON. If EPSILON is less than RESPECT, then the imperfection calculation is skipped for that local direction, for that case, for that member. The imperfection calculation is also skipped for any member that is in tension.

\[
\text{EPSILON}_Y = \text{Length} \times \sqrt{\frac{\text{(abs(axial load))}}{\text{EI}_Z}} \\
\text{EPSILON}_Z = \text{Length} \times \sqrt{\frac{\text{(abs(axial load))}}{\text{EI}_Y}}
\]

**Related Links**

- [M. To assign cracked section properties to a member](on page 802)
- [M. To add a member specification](on page 797)
- [M. To assign nonlinear cable members](on page 801)
- [G.8.1 Truss and Tension- or Compression-Only Members](on page 2330)
- [M. To assign axial action members](on page 798)
- [G.7 Member and Element Release](on page 2329)
- [M. To assign member end release](on page 799)
- [G.11 Member Offsets](on page 2334)
- [M. To assign member end offsets](on page 800)
- [M. To assign member imperfection for members](on page 800)
Ribbon Control Reference
Specification tab

- *M. To assign cracked section properties to a member* (on page 802)
- *M. To assign member fire proofing* (on page 803)
- *EX. US-4 Inactive Members in a Braced Frame* (on page 4390)
- *EX. UK-4 Inactive Members in a Braced Frame* (on page 4670)

**Plate Specs dialog**
Used to specify finite element specifications.
Opens when Plate is clicked on the Specifications - Whole Structure dialog.
This dialog contains multiple tabs, each of which is used to add a different member specification.

**Release tab**
Used to release one or more degrees of freedom at the corner nodes of the elements. Unless a release is specified, elements are rigidly connected at the nodes (all six degrees of freedom are restrained) unless the node is a cantilever node or is supported.

**Note:** Refer to *TR.22.2 Element Release Specification* (on page 2490) for additional information on plate element releases.

- **Node** Select the node to be released. The node number depends on the Element Incidence for the element.
- **Release** Check the boxes for the FX, FY, FZ, MX, MY, or MZ directions to define the element release condition. These are based on the local coordinate system for the element.

**Ignore Inplane Rotation tab**
Used to override the value that STAAD determines for the stiffness associated with the local axis Mz degree of freedom for plate elements.
Refer to *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2498) for additional information.
There are no additional parameters for this plate element specification.

**Rigid Inplane Rotation tab**
Used to program to connect the corner Mz "in-plane rotation" action to the other corner Mz rotations rigidly.
Refer to *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2498) for additional information.
There are no additional parameters for this plate element specification.
Plane Stress tab
Used to model plate elements as Plane Stress elements.
Refer to TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2498) for additional information.
There are no additional parameters for this plate element specification.

Ignore Stiffness tab
Used to instruct the program to neglect the stiffness of the selected plates while assembling the stiffness matrix. Hence, the stiffness contribution of the selected plates is not considered. This is similar to the Inactive command for the members.

When modeling plate elements, often we come across situations where we want the plates to carry loads, but at the same time do not want the stiffness of the plates to be considered in the analysis. For example, structural units such as glass panels or corrugated sheet roofs are subjected to loads like wind pressures or snow loads. While these elements are designed to carry the loads and transmit the same to the other parts of the structure, they are not designed to provide any additional stiffness to the structure.

Note: Refer to TR.22.3 Element Ignore Stiffness (on page 2491) for additional information.
There are no additional parameters for this plate element specification.

Related Links
• G.7 Member and Element Release (on page 2329)
• M. To assign plate corner release (on page 807)
• G.5.1 Plate and Shell Elements (on page 2308)
• M. To assign plates as plane stress (on page 808)
• G.5.1 Plate and Shell Elements (on page 2308)
• M. To assign inplane rotation behavior to plates (on page 809)
• G.5.1 Plate and Shell Elements (on page 2308)
• M. To ignore plate stiffness (on page 810)

Properties - Whole Structure dialog
Used to define and assign member, plate element and material properties.

Section tab
Used for defining member section properties and element thickness and assigning them to members and elements. Properties can be defined and assigned to members and elements using the options available on the Section tab. As property values are assigned, the Section Tab is updated with their tag number and property description. The items on the Section tab are described below.

Beta Angle tab
Used to define beta angles (rotation of the member about its local x-axis) and assign them to members.

Related Links
• G.6.2 Built-In Steel Section Libraries (on page 2325)
Section Profile Tables dialog

Used to assign catalog sections for steel, timber, and aluminum members. Sections or organized by material, country, and then profile.

Opens when Section Database is clicked on the Properties - Whole Structure dialog.

**Tables list** Select the type of section by clicking on the appropriate tab and then select the specific section from the list box. Please note that depending on the type of section selected, additional properties may be specified.

**View Table** This option displays all member properties for the aluminum table.

**Material** Select this option to select the material from the drop down list if the new member property tag should include the material constants.

**Type Specification** Refer to TR.20.1 Assigning Properties from Steel Tables (on page 2461) for details on type specifications for different table profiles.

Steel tab

Used to select hot-rolled and joint sections. Displays the contents of the various country steel tables in a series of tabs on the left hand side of the dialog.

Coldformed Steel tab

Used to select cold-formed steel sections.

Timber tab

Used to select standard species and size timbers as well as glue-laminated sections for American and Canadian standards.

**Naming Convention** Click and hold to display a pop-up of the nomenclature conventions used for the currently selected catalog.

Aluminum tab

Used to assign aluminum sections to members from the built-in American aluminum table.

Related Links

- TR.31.2.14 IBC 2015 Seismic Load Definition
- G.6.2 Built-In Steel Section Libraries (on page 2325)
- M. To add a new table section property (on page 730)
- M. To add an American steel joist section (on page 731)
Property dialog

Used to assign prismatic cross sections to members. Opens when the Beam Profiles > Prismatic Profiles tool is selected in the Specifications group on the Specification ribbon tab.

Note: The Properties - Whole Structure dialog also opens simultaneously so that some of the other options available from that dialog may be utilized.

Circle tab

**YD**  
Section diameter.

**Material**  
Check this box and select the material from the drop down list if the new member property tag should include the material constants.

Rectangle tab

**YD**  
Depth of the member in local y direction.

**ZD**  
Width of the member in the local z direction.
Tee tab

**YD**  Depth of the member in local y direction.

**ZD**  Width of the flange.

**YB**  Depth of the stem (web).

**ZB**  Width of the stem (web).

Trapezoidal tab

**YD**  Depth of the member in local y direction.

**ZD**  Top width.

**ZB**  Bottom width.

General tab

**AX**  Cross sectional area of the member.

**AY**  Effective shear area in local y-axis.

**AZ**  Effective shear area in local z-axis.

**IX**  Torsional constant.

**IY**  Moment of inertia about local y-axis.

**IZ**  Moment of inertia about local z-axis (usually major).

**YD**  Depth of the member in local y direction. Used as the diameter of section for circular members.

**ZD**  Depth of the member in local z direction. If ZD is not provided and YD is provided, the section will be assumed to be circular.

**YB**  Depth of stem for T-section.

**ZB**  Width of stem for Tee section or bottom width for trapezoidal section.

Tapered I tab

**F1**  Depth of section at start node.

**F2**  Thickness of web.

**F3**  Depth of section at end node.

**Note:** F3 should be *less than* F1 (i.e., the section should decrease in depth from start to end). You must provide the member incidences accordingly.

**F4**  Width of top flange.

**F5**  Thickness of top flange.

**F6**  Width of bottom flange. Defaults to F4 is zero.

**F7**  Thickness of bottom flange. Defaults to F5 if zero.
Tapered Tube tab

**Type of Section**
Select one of the profile shapes:
- Round
- Hexagonal – 16-sided
- Dodecagonal – 12-sided
- Octagonal – 8-sided
- Hexagonal – 6-sided
- Square – 4-sided

**d1**
Depth of section at start of member.

**d2**
Depth of section at end of member.

**th**
Thickness of section (constant throughout the member length).

**Notes:**
1. Section properties are calculated using the rules applicable for thin-walled sections.
2. Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the `SET SHEAR` command has no effect on the deformation computed for members with these cross sections.

Assign Profile tab

Used to instruct the program to select a suitable steel section based on a profile classification, such as beam, column, double-angle, etc.

**Select Profile Specification**
Specify a profile by selecting one of the options:
- Angle
- Double Angle
- Beam
- Column
- Channel

The program will then assign a section based on that profile from the relevant built-in Steel table.

**Related Links**
- *G.6.1 Prismatic Properties* (on page 2323)
- *M. To assign a prismatic section* (on page 733)

**Property: Tapered I dialog**

Used to specify an I-section having a varying depth over the length of the member.

Opens when the **Beam Profiles > Tapered Profile** tool is selected in the **Specifications** group on the **Specification** ribbon tab.
Note: This dialog is identical to the Tapered I tab of the Property dialog.

F1  Depth of section at start node.
F2  Thickness of web.
F3  Depth of section at end node.

Note: F3 should be less than F1 (i.e., the section should decrease in depth from start to end). You must provide the member incidences accordingly.

F4  Width of top flange.
F5  Thickness of top flange.
F6  Width of bottom flange. Defaults to F4 is zero.
F7  Thickness of bottom flange. Defaults to F5 if zero.

Related Links
• G.6.4 Tapered Sections (on page 2326)
• M. To assign a tapered I section (on page 734)

User Provided Table dialog

Used for adding and assigning previously created user table sections to the structure.

Note: The Properties - Whole Structure dialog (on page 2968) also opens simultaneously so that some of the other options available from that dialog box may be utilized.

Opens when the Beam Specifications > User Table Profile tool is selected in the Specifications group on the Specification ribbon tab.

Select Existing Table  Select a previously created User Table number from the drop down list. User Tables are created using the Create User Provided Table dialog (on page 2974).
Section List  Select the specific section from the list.
Material  Check this box and select the material from the drop down list if the new member property tag should include the material constants.

Add  Click to add this property to the structure.
Assign  Click to assign the property to selected members as well as add this property to the structure.
Close  Closes the dialog without adding any properties.
Help  Opens the STAAD.Pro Help window.

Related Links
• G.6.3 User-Provided Steel Table (on page 2325)
• M. To create a general section (on page 735)
**Create User Provided Table** dialog

Used to add new or edit existing user provided section properties tables.

Opens when either:

- you select a section type and click **OK** in the **Select Section Type** dialog, or
- or when you select **Beam Profiles > User Table Profile** tool in the **Specifications** group on the **Specification** tab, but no user provided tables are present.

![User Table Manager](image)

**Select Existing Table**

If you have already created a user provided table, you may select it from this drop-down list box for further editing.

**Note:** If you select an external table (indicated by “(EXT)”), then the file path and file name will be displayed below.

**New Table**

Opens the **New User Table** dialog, which is used to select the section type for a user provided table and to optional specify an external table file.

**Add**

Click this to specify a custom UPT section after creating a new table or selecting an existing table. A dialog opens for inputting property values applicable to the section type chosen for the current table.

**Tip:** After entering the section dimensions, you can click **Calculate** to have the program calculate the derived section properties (areas, moments of inertia, etc.).

Once the section is defined, it may be assigned from the **Properties - Whole Structure** dialog.

**Note:** You may not add entries to external tables from this interface.

**Edit / View**

Click to open the details of a selected table data entry for viewing or editing.
**Delete**
Click to delete the currently selected table data entry from the table.

**Delete Table**
Deletes the current user provided table.

**Save Table**
Saves changes made to the user provided table.

**Close**
Closes the dialog.

**Section Wizard**
(Displayed only for External tables) Click to open the Section Wizard interface.

**Related Links**
- G.6.3 User-Provided Steel Table (on page 2325)
- M. To create a general section (on page 735)

**New User Table** dialog

Used to select the section type for a User Provided Table and to optional specify an external table file.

Opens when New Table is selected in the User Table Manager dialog.

*External Table*  
Set this option to load section data from an external file (e.g., a .upt file exported from Section Wizard). The File Name must be provided.

*File Name*
Type a name for the external table file. Click **Browse** to search for a location from which to load the table file.

*Select Section Type*
Select the type of cross section contain in the table. All sections within a table must be of the same type.
- Wide Flange
- Channel
- Angle
- Double Angle
- Tee
- Pipe
- Tube
- General
- I Section
- Prismatic
**Wide Flange** dialog

Used to specify the section properties for a wide flange user defined table section.

Opens when you click **Add New Property** in the **Create User Provided Table** dialog when a Section Type: Wide Flange table is selected.

| **Note:** Refer to **Section 5.19.1 of the Technical Reference Manual** (on page 2449) for detailed descriptions on properties of Wide Flange sections in User Steel tables. |

**Section Name** If you have already created a User Provided Table, you may select it from this drop-down list box for further editing.

| **D** | Depth of the section |
| **TF** | Thickness of top flange (or both flanges when WF1 is not specified) |
| **WF** | Width of the top flange (or both flanges when WF1 is not specified) |
| **TW** | Thickness of web |
| **TF1** | Thickness of bottom flange |
| **WF1** | Width of the bottom flange |
| **Cross Section Area (Ax)** | Cross section area. Can be automatically calculated by clicking the **Calculate** button. |
| **Inertia About Local z (Iz)** | Moment of inertia about local z-axis (usually strong axis). Can be automatically calculated by clicking the **Calculate** button. |
| **Inertia About Local y (Iy)** | Moment of inertia about local y-axis. Can be automatically calculated by clicking the **Calculate** button. |
| **Torsional Constant (Ix)** | Moment of inertia about local x-axis. Can be automatically calculated by clicking the **Calculate** button. |
| **Shear Area in Y (Ay)** | Shear area in local y-axis. If zero, shear deformation is ignored in the analysis. Can be automatically calculated by clicking the **Calculate** button. |
| **Shear Area in Z (Az)** | Same as above except in local z-axis. Can be automatically calculated by clicking the **Calculate** button. |
| **Calculate** | Click to automatically calculate the Ax, Iz, Iy, Ix, Ay, and Az values based on the dimensions. |
| **Additional Composite Flange** | Set this option to add a composite concrete slab to the wide flange section. |
| **B(left)** | Width of the composite slab to the left of the web center line |
| **B(right)** | Width of the composite slab to the right of the web center line |
Ribbon Control Reference
Specification tab

**Thickness**  Thickness of the composite slab

**Modular Ratio**  The ratio of the modulus of elasticity of steel to concrete

**Additional Bottom Steel Plate**
Set this option to add a bottom cover plate to the wide flange section. This may only be included if the **Additional Composite Flange** option is selected.

- **B(left)**  Width of the additional bottom flange plate to the left of the web center line
- **B(right)**  Width of the additional bottom flange plate to the right of the web center line
- **Thickness**  Thickness of the additional bottom flange plate

**OK**  Saves the UPT section and closes the dialog.

**Cancel**  Closes the dialog without creating a new section.

**Material Constant dialog**

Used to specify individual material constants of the materials of which members and elements are comprised.

Opens when a constant name is selected from the **Constants** tool in the **Materials** group on the **Specifications** ribbon tab. A separate dialog is used to define each constant value.

The types of constants available to define are:

- **Young's Modulus**  – the modulus of elasticity, $E$
- **Poisson’s Ratio**, $\nu$
- **Shear Modulus**, $G$
- **Density**, $\gamma$
- **Thermal Coefficient**, $\alpha$
- **Damping Ratio**, $c$

**Material Constant**
Select the pre-defined materials Aluminum, Concrete, or Steel to use a built-in value. See below for the built-in material values.

Alternately, select the **Enter Value** option to type in a custom constant value in the current units.

**Assign**
Select the scope of members which the material constant is to be assigned.

- **To View**  assigns the material constant to all members within the current view.
- **To Selection**  assigns the material constant to only those members selected.

**Built-In Material Constants**

For $E$, $G$, POISSON, DENSITY, ALPHA, and CDAMP, built-in material names can be entered instead of a value for $f_l$. The built-in names are **STEEL**, **CONCRETE**, & **ALUMINUM**. Appropriate values will be automatically assigned for the built-in names.
Table 310: Constants (in Kip, inch, Fahrenheit units)

<table>
<thead>
<tr>
<th>Constant</th>
<th>Material</th>
<th>Steel</th>
<th>Concrete</th>
<th>Aluminum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>E (US)</td>
<td>Steel</td>
<td>29,000</td>
<td>3,150</td>
<td>10,000</td>
<td>Kip/in²</td>
</tr>
<tr>
<td>Poisson’s</td>
<td>Concrete</td>
<td>.30</td>
<td>.17</td>
<td>.33</td>
<td>(ratio)</td>
</tr>
<tr>
<td>Density</td>
<td>Aluminum</td>
<td>.000283</td>
<td>.000868</td>
<td>.000098</td>
<td>Kip/in³</td>
</tr>
<tr>
<td>Alpha</td>
<td>Steel</td>
<td>6.5E-6</td>
<td>5.5E-6</td>
<td>12.8E-6</td>
<td>L/L/° F</td>
</tr>
<tr>
<td>CDAMP</td>
<td>Concrete</td>
<td>.03</td>
<td>.05</td>
<td>.03</td>
<td>(ratio)</td>
</tr>
<tr>
<td>E (nonUS)</td>
<td>Steel</td>
<td>29,732.736</td>
<td></td>
<td></td>
<td>Kip/in²</td>
</tr>
</tbody>
</table>

Table 311: Constants (in MKS, Celsius units)

<table>
<thead>
<tr>
<th>Constant</th>
<th>Material</th>
<th>Steel</th>
<th>Concrete</th>
<th>Aluminum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>E (US)</td>
<td>Steel</td>
<td>199,947,960</td>
<td>21,718,455</td>
<td>68,947,573</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Poisson’s</td>
<td>Concrete</td>
<td>.30</td>
<td>.17</td>
<td>.33</td>
<td>(ratio)</td>
</tr>
<tr>
<td>Density</td>
<td>Aluminum</td>
<td>76.819 541</td>
<td>23.561612</td>
<td>26.601820</td>
<td>kN/m³</td>
</tr>
<tr>
<td>Alpha</td>
<td>Steel</td>
<td>12.0E-6</td>
<td>10.0E-6</td>
<td>23.0E-6</td>
<td>L/L/° C</td>
</tr>
<tr>
<td>CDAMP</td>
<td>Concrete</td>
<td>.03</td>
<td>.05</td>
<td>.03</td>
<td>(ratio)</td>
</tr>
<tr>
<td>E (nonUS)</td>
<td>Steel</td>
<td>205,000,000</td>
<td></td>
<td></td>
<td>kN/m²</td>
</tr>
</tbody>
</table>

**Note:** E (US) is used if US codes were installed or if Member Properties American is specified for an analysis; otherwise E (nonUS) is used.

**Related Links**
- M. To assign material constants (on page 793)
- G.17.3.3.1 Composite Damping (on page 2367)
- M. To assign a composite damping ratio (on page 874)

**Beta Angle** dialog

Used to specify the Beta angle for members.

Opens when:
- the Beam >Beta Angle tool is selected in the Specifications group on the Specification ribbon tab, or
• **Create Beta Angle** is clicked on the **Properties - Whole Structure** dialog **Beta Angle** tab

![Beta Angle dialog](image)

- **Angle** Type the rotation about the longitudinal member axis (i.e., local x axis) in the **Angle in Degrees** field.

  The **Angle** and **RAngle** options are for use only with single angle sections. These options are further described in **G.4.3 Relationship Between Global and Local Coordinates** (on page 2301).

- **Assign** Select the scope of members which the geometric constant is to be assigned.
  - **To View** assigns the geometric constant to all members within the current view.
  - **To Selection** assigns the geometric constant to only those members selected.

**Related Links**
- **G.4.3 Relationship Between Global and Local Coordinates** (on page 2301)
- **M. To assign a member rotation angle** (on page 794)
- **M. To align a single angle to its flanges** (on page 795)

**Reference Point** dialog

Used to specify a reference point for the members from which the program calculates the beta angle.

Opens when the **Beam Specifications > Beam Reference Point** tool is selected from the **Specifications** group on the **Specification** ribbon tab.
**Point**  Provide the global coordinates of the point towards which the minor axis (local y axis) of the member should be directed. The point should not lie on the longitudinal axis (local x axis) of the member.

**Vector**  A vector can be assigned with respect to the local coordinate system of the member. The local Y axis for the member is going to oriented along this vector.

This can be used to orient a member without the need to calculate a beta angle. For example, vector values of $X = 0$, $Y = 2$, and $Z = 1$ will create a vector at a slope of 1:2 from the member's axis. The resulting beta angle $= \tan^{-1}(1/2) = 26.5651^\circ$.

Refer to [AD.2006.1.6 Specification of Member Orientation Using Reference Vector](on page 324) section for additional information on this feature.

**Note:** The Member Query dialog Geometry tab will report the actual beta angle used.

**Node**  An existing node in the structure can be set as the reference point for one or more members. This is the alternative to defining the coordinates of the reference point. When this button is turned on, the list of all nodes in the structure will appear in the drop-down list. Select the desired node number.

**Assign**  Select the scope of members which the geometric constant is to be assigned.

- **To View** assigns the geometric constant to all members within the current view.
- **To Selection** assigns the geometric constant to only those members selected.

**Related Links**

- [G.4.3 Relationship Between Global and Local Coordinates](on page 2301)
- [M. To align a member to a reference point](on page 796)
Plate Element Property dialog

Used to provide plate element properties (thickness) with or without the material specification. Plates can have a different thickness at each node.

Opens when:

- the Plate Thickness tool is selected from the Plate Profiles group on the Specification ribbon tab, or

- Thickness is clicked on the Properties - Whole Structure dialog

**Node 1, Node 2, etc.** Enter the thickness at each of the nodes. For plates of uniform thickness, entering the thickness at Node 1 only will suffice. The other three values will default to that of Node 1 if no input is provided for them.

**Material** Check this box and select the material from the drop down list if the new element property tag should include the material constants. The specification of the standard materials (that is, Concrete, Steel, or Aluminum) can be added in this dialog box or can be done separately with other material specifications.

Related Links
- G.5.1 Plate and Shell Elements (on page 2308)
- M. To specify plate thickness (on page 806)

Plate Reference Point dialog

Used to specify options for determining the general direction of the local Z axis of elements.

Opens when the Plate Specifications > Plate Reference Point tool is selected from the Specifications group on the Specification ribbon tab.
**Point**  Type the global coordinates of a point to use for orienting the local z axis of the plate elements.

**Note:** The local coordinate system of the plate must still obey that the xy plane lie in the plane of the element itself. So the z axis won’t necessarily point directly to this coordinate. Rather, the z axis will point away from the element to the side that the reference point is located.

**Local Z Axis**  Select an option to define the direction of the local z axis of the plate element with reference to the point provided.

**Assign**  Select the scope of plates which the geometric constant is to be assigned:

- **To View** – assigns the plate reference point to all plate elements within the current view.
- **To Selection** – assigns the plate reference point to only those plate elements selected.

**Related Links**

- [G.5.1 Plate and Shell Elements](#) (on page 2308)
- [M. To align a plate to a reference point](#) (on page 805)

## Supports - Whole Structure dialog

Used to define supports and assign them to nodes. All supports that have been defined for the model are listed here.

**Edit**  Click to edit the parameters of the one of the following support types after it is created.

- Fixed But
- Enforced But
- Inclined
- Foundation

**Note:** You cannot edit support types which are defined, such as fixed, pinned, or enforced.

**Create**  Opens the **Create Support** dialog opens, which is used to create various support types.

**Delete**  Click to delete a previously assigned support.

**Assignment Method**  The options under the Assignment Method offer different choices for assigning supports to the structure.

**Assign To Selected Nodes**  To assign a support to selected nodes, first select the support from the Supports dialog box. The support selected is highlighted. Then select the nodes to which this support is to be assigned. When all desired nodes are selected, click the Assign To Selected Nodes radio button, then click the Assign button.

**Assign To View**  To assign a support to all free nodes in a view, first select the support from the Supports dialog box. The selected support is highlighted. Select the Assign To View radio button, then click the Assign button. All free nodes in the structure are assigned this support.

**Use Cursor To Assign**  To assign a support to nodes using the cursor, first select the support from the Supports dialog box. The selected support is highlighted. Select the Use...
Cursor To Assign radio button, then click the Assign button. The button will appear depressed and the label will change to Assigning. Make sure that the Nodes Cursor is selected so that we can select the nodes. Using the cursor, click on the nodes to which this support is to be assigned. Click on the Assign button again to finish.

**Assign To Edit List**

To assign a support using a typed list of node numbers, first select the support from the Supports dialog. The selected support is highlighted. Select the Assign To Edit List radio button, then type the list of node numbers and click the Assign button.

**Related Links**

- *M. To assign a fixed or pinned support* (on page 812)
- *G.13 Supports* (on page 2335)

**Create Support** dialog

Used to define support (boundary) conditions of the structure to nodes. The dialog consists of a series of tabs, each of which are used to assign a different support condition. Each tab includes options for the specified support condition.

**Note:** The Supports - Whole Structure dialog also opens simultaneously so that some of the other options available from that dialog box may be utilized.

If the support is not assigned as it is created, it can be assigned later from the Support dialog. However, note that if it is not assigned to at least one spring support when is closed, then the definition will not be saved in the STD file.

**Fixed tab**

Used to create a fixed support tag and (optionally) to assign that to selected nodes. A Fixed support is restrained in all six degrees of freedom.

Refer to *TR.27.1 Global Support Specification* (on page 2514) for additional information on global supports.

Opens when Add Support > Fixed is selected from the Specification ribbon tab Supports group.

There are no settings for assigning a Fixed support.

**Pinned tab**

Used to create a pinned support tag and (optionally) to assign it to selected nodes. A Pinned support is restrained in all three translational degrees of freedom and free in the 3 rotational degrees of freedom.

Refer to *TR.27.1 Global Support Specification* (on page 2514) for additional information on global supports.

Opens when Add Support > Pinned is selected from the Specification ribbon tab Supports group.

There are no settings for assigning a Pinned support.
**Fixed But tab**

Used to create various types of roller, hinge, and spring supports with specified restrained degrees of freedom and to assign them to selected nodes.

Refer to TR.27.1 Global Support Specification (on page 2514) for additional information on global supports.

Opens when Add Support > Fixed But/Spring is selected from the Specification ribbon tab Supports group.

- **Release** Select the degrees of freedom you want to release.
- **Define Spring** To specify a spring support, the spring constant in the appropriate edit box.

**Enforced tab**

Used to create a fixed-condition support definition. The Enforced support is the same as a Fixed support except that the restrained degrees of freedom are defined in terms of being stiff springs. The real advantage of using the Enforced type lies in the fact that it enables STAAD.Pro to accept loads such as support displacement loads in the case of plates and solids. Support displacement loads are not permitted for plates and solids if the FIXED support type is used. So, for structures without these characteristics, the Fixed type of support offers the same level of functionality as the Enforced support type.

Refer to TR.27.1 Global Support Specification (on page 2514) for additional information on global supports.

There are no settings for assigning an Enforced support.

**Enforced But tab**

Used to create a fixed-condition support defined in terms of being stiff springs with the option to release selected degrees of freedom. The Enforced But support type is the same as the Enforced support except that we have a choice on the degrees of freedom we wish to restrain. For example, you can select Enforced But and restrain just the FX, FY, and FZ degrees of freedom, and let the remaining three free to deform.

Refer to TR.27.1 Global Support Specification (on page 2514) for additional information on global supports.

- **Release** Select the degrees of freedom you want to release.

**Inclined tab**

Used to create supports that offer restraints in an axis system that is inclined with respect to the global axis system.

Refer to TR.27.2 Inclined Support Specification (on page 2516) for additional details on specifying inclined supports.

- **Incline Reference Point** Specify a point in space to define the incline.
- **Support Type** Select to use one of the standard STAAD.Pro support types: Pinned, Fixed, Fixed But, Enforced, or Enforced But.
### Release

(Fixed But or Enforced But options only) Select the forces and moments to release by checking the respective box.

### Spring

(Fixed But option only) To specify a spring support, the spring constant in the appropriate edit box.

---

**Foundation tab**

Used to create spring supports for independent footings and mat foundations and to assign them to selected nodes.

Refer to [TR.27.3 Automatic Spring Support Generator for Foundations](on page 2517) for additional details on Foundation supports.

**Foundation**

- **Footing** - Select this option to define a spring support for an isolated footing. Provide the Length (\(L\)) and Width (\(W\)) dimensions of the footing in current units.

  In generating spring supports for mat foundations, there are two methods available in STAAD.Pro. Both those options require the program to calculate the influence area of the nodes which define the surface, and then multiply that area by the subgrade modulus of the medium. The difference between these two options lies in the way the influence area is calculated.

  - **Elastic Mat** - In this method, the area is calculated using a Delaunay triangle principle. Hence, the candidates for this option are the nodes which define the mat. To achieve best results, one needs to ensure that the contour formed by the nodes form a convex hull.

  - **Plate Mat** - If the foundation slab is modeled using plate elements, the spring supports can be generated using an influence area calculated using the principles used in determining the tributary area of nodes from the finite element modeling standpoint. Hence, the candidates for this option are the plates which define the mat. When the mat is modeled using plates, this produces superior results than the ELASTIC MAT type.

**Direction**

The \(X, Y, Z, XONLY, YONLY, \) and \(ZONLY\) indicate the direction of resistance of the spring supports. If \(X, Y\) or \(Z\) is selected, then a spring support is generated in that direction only whereas the associated rotational degree of freedom and the other two translational d.o.f receive a fixed support. For example, if \(Y\) is selected, then \(FY\) is supported by a spring support, where as \(MY, FX\) and \(FZ\) are fixed supports; and \(MX\) and \(MZ\) are free. If \(XONLY, YONLY, \) or \(ZONLY\) is selected, then a spring support in that direction alone is generated, and every other d.o.f is set to be free to deform.

**Subgrade**

Type a value for the subgrade modulus of the soil.

**Print Influence Area of Each Joint**

(For Mat footing options only) Select this option to have the influence area of each support included in the output. The are used in the calculation of the spring stiffness of each joint used when defining a Plate Mat or Elastic Mat command will be included.

**Generate Compression Only/Multi-linear Spring**

(For Mat footing options only) When the Compression Only option is set, then if after any of the cycles of analysis, the force at a node included in the command range (in the elastic mat range or used to define a plate in the plate mat range) is found to be tensile (i.e. negative reaction), then the load case is marked for a re-analysis with that support removed.
**Multilinear Spring tab**

Used to enter displacement and spring stiffness data pairs for a multilinear spring definition, which is then used for a Fixed But support type.

The Multi Linear support type allows the user to model the support type for which the resistance offered to external loads varies with the extent of deformation of the support node. For example, when its behavior in tension differs from its behavior in compression, as in the case of soil springs, this facility can be utilized. Another example is a partial roller support where translation can occur without any resistance for a certain amount of displacement, after which it becomes fully restrained. The problem is solved iteratively using cycles of analysis and convergence checks. Hence, only on static load case can be specified per analysis command. It cannot be used with dynamic load cases.

Refer to TR.27.4 Multilinear Spring Support Specification (on page 2520) manual for additional information.

### Multilinear Spring table

Values are entered in order from least to greatest displacement (including negative displacements, which are then ordered first) and their corresponding spring stiffness values.

---

**Tension/Compression Only Spring tab**

Used for support types which are capable of unidirectional action only, such as soil under a foundation slab.

Refer to TR.27.5 Spring Tension/Compression Specification (on page 2522) for additional details on Tension/Compression Only Spring supports.

### Reaction Type

Select whether the support degree(s) of freedom (d.o.f) are tension only or compression only. Indicates that if, after any of the cycles of analysis, the direction of the force in the spring is of the wrong ‘type’, then the support will be removed from that direction and a new analysis performed.

<table>
<thead>
<tr>
<th>Reaction Type</th>
<th>Support will be active if the reaction is...</th>
<th>Support will be made inactive and a new analysis will be performed if the reaction is...</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tension Only</td>
<td>- ve</td>
<td>+ ve</td>
</tr>
<tr>
<td>Compression Only</td>
<td>+ ve</td>
<td>- ve</td>
</tr>
</tbody>
</table>

### Spring Direction

Select the degree(s) of freedom to set as unidirectional. More than one d.o.f may be set at a time.

**Related Links**

- [M. To assign a fixed or pinned support](on page 812)
- [G.13 Supports](on page 2335)
- [M. To assign a foundation support](on page 818)
- [G.13 Supports](on page 2335)
- [EX. US-23 Spring Support Generation for a Slab on Grade](on page 4541)
- [EX. UK-23 Spring Support Generation for a Slab on Grade](on page 4822)
- [M. To assign an enforced support](on page 813)
Material - Whole Structure dialog box

Used to define, manage, and assign materials used in the STAAD.Pro input file. The Isotropic tab is used for defining isotropic materials and assigning them to members and elements. The Orthotropic 2D tab is used for defining materials with different properties in two dimensions and assigning them to members and elements.

Material list
All materials that have been defined for the model are listed here. Select the Isotropic or Orthotropic 2D tab to display materials which exhibit those directional properties.

We may specify Materials in two ways. We may first create a material tag and then select the members or elements to which this tag is applied. Alternatively, we may first select the members or elements and then specify a material to be assigned to the selected items. In the second case, a new material specification is created along with a material tag. Please note that the Assign button becomes active if you have already selected the members or elements to which the material tag is to be applied.

Highlight Assigned Geometry
When this box is checked and we click on a material tag in the Material dialog box, members that have been assigned the material tag will appear highlighted in the structure window. For example in the previous figure, if we click on the STEEL material tag, all steel members will be highlighted.

Create
When the Isotropic tab is selected, clicking Create opens the Isotropic Material dialog (on page 2989), which is used to add a new isotropic material and properties to the STAAD.Pro input file. When the Orthotropic 2D tab is selected, this button opens the 2D Orthotropic Material Property dialog (on page 2989), which is used.

Edit
Opens the Isotropic Material dialog (on page 2989) or the 2D Orthotropic Material Property dialog (on page 2989), which can be used to edit the properties for the currently selected material (the name field is inactive when editing).

Delete
Deletes the currently selected material from the list.

Tip: If you accidentally delete one of the predefined materials, they can still be added using the Create button and selecting the predefined material name from the drop-down list.

Assignment Method
The options under the Assignment Method are used to assign material tags to members and elements. First select a material tag from the Materials dialog box. Use one of the following
assignment methods to apply this material: Assign To Selected Beams, Assign To View, Use Cursor To Assign or Assign To Edit List.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assign To Selected Beams</td>
<td>To assign a material to selected members or elements, first select the material tag from the Materials dialog box. The selected material is highlighted. Then select the members or elements to which this material is to be assigned. This is done by going to the Select menu, then choosing the appropriate cursor option. Select the desired members or elements using the cursor. When all desired geometry is selected, click the Assign To Selected Beams radio button, then click the Assign button. Note that the label for this button changes depending on whether members or elements are selected.</td>
</tr>
<tr>
<td>Assign To View</td>
<td>To assign a material to all members or elements in a view, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To View radio button, then click the Assign button. All members or elements in the structure are assigned this material.</td>
</tr>
<tr>
<td>Use Cursor To Assign</td>
<td>To assign a material to members or elements using the cursor, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Use Cursor To Assign radio button, then click the Assign button. The button will appear depressed and the label will change to Assigning. Use the appropriate cursor (from the Select menu) to assign the selected material to the individual members or elements. Click on the Assign button again to finish.</td>
</tr>
<tr>
<td>Assign To Edit List</td>
<td>To assign a material using a typed list of member or element numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button.</td>
</tr>
</tbody>
</table>
Assigns the material per the selected assignment method.

Closes the dialog.

**Isotropic Material dialog**

Used to add a new isotropic material and properties to the STAAD.Pro input file.

**Note:** Steel, Concrete, Stainless Steel, and Aluminum are default materials in the program and are always included.

- **Title**
  Enter a unique name to use for a new material. The drop-down list provides the predefined materials available in STAAD.Pro.

- **Material Properties**
  Define each of the material properties:
  - Young's Modulus (E)
  - Poisson's Ratio (nu)
  - Density
  - Thermal Coefficient (a)
  - Critical Damping
  - Shear Modulus (G)

- **Type of Material**
  Select a predefined material type for associating Design Properties with the material. If one of the predefined materials is not appropriate, select Not Specified and no Design Properties will be assigned.

- **Yield Stress (Fy)** (Steel only)
- **Tensile Strength (Fu)** (Steel only)
- **Yield Strength Ratio (Ry)** (Steel only)
- **Tensile Strength Ratio (Rt)** (Steel only)
- **Compressive strength (Fcu)** (Concrete only)

**Related Links**
- [M. To create a material definition](on page 788)

**2D Orthotropic Material Property dialog**

Used to add a new orthotropic material and properties to the STAAD.Pro input file.

Refer to [TR.26.1 Define Material](on page 2501) for additional information.

- **Title**
  Enter a unique name to use for a new material. The drop-down list provides the predefined materials available in STAAD.Pro.

- **Material Properties**
  Define each of the material properties for the local X and Y directions:
  - **Young's Modulus** (E) – modulus of elasticity
  - **Thermal Coefficient** (a)

The general properties are not orthotropic:
• Density
• Critical Damping
• Poisson’s Ratio (v)

The Shear Modulus (G) property is defined as follows:
• Gxy – in-plane shear
• Gyz – shear transverse to the local yz plane
• Gxz – shear transverse to the local xz plane

Related Links
• M. To create an orthotropic material (on page 791)

Section Database Manager window

Used to view or edit the existing steel databases by country code and section type. On selection of the section type, the adjacent list box displays the available sections defined in the database. Selection of section in the list box enables the buttons for operating on the database.

Opens when the Section > Section Database tool is selected in the Specifications group on the Specification ribbon tab.
Double-clicking on a section opens a new dialog where there are buttons for operating on the database.

Table 312: SectionDBManager Home ribbon tab

<table>
<thead>
<tr>
<th>Tool</th>
<th>What it Does</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close Table</td>
<td>Closes the currently selected table.</td>
</tr>
<tr>
<td>Close All</td>
<td>Closes all open tables.</td>
</tr>
<tr>
<td>Print</td>
<td>Click to open the currently displayed table in a Print Preview window, where you can print the contents to a Windows printer.</td>
</tr>
<tr>
<td>Export</td>
<td>Click to export the currently displayed table as a Microsoft® Office Excel® spreadsheet (file extension .xlsx).</td>
</tr>
<tr>
<td>Lock / Unlock</td>
<td>Click to unlock the table for editing. Editing-related tools (e.g., Append Row) are made active when unlocked. Click again to lock the table against further editing.</td>
</tr>
<tr>
<td>Tool</td>
<td>What it Does</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Append Row</td>
<td>Adds a new section to the database under the selected country and section type. It displays the following dialog box for the input of section properties.</td>
</tr>
<tr>
<td>Add Above</td>
<td>Click to add a new record before the currently selected row.</td>
</tr>
<tr>
<td>Add Below</td>
<td>Click to add a new record after the currently selected row.</td>
</tr>
<tr>
<td>Delete Row</td>
<td>Deletes the selected section in the list box from the database under the selected country and section type.</td>
</tr>
<tr>
<td>Commit</td>
<td>Saves changes made to the current table.</td>
</tr>
<tr>
<td>Rollback</td>
<td>Undoes any changes made to the current table.</td>
</tr>
<tr>
<td>Font controls</td>
<td>Use to change the display font of tables.</td>
</tr>
<tr>
<td>Search</td>
<td>Use to filter the table for the typed string. Click the “X” to clear the filter and display the entire table again.</td>
</tr>
</tbody>
</table>

### Loading tab

**Table 313: Loading Specifications group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Load Cases</td>
<td>Opens the <strong>Create Primary Load Case</strong> dialog, which is used to create new primary load cases.</td>
</tr>
</tbody>
</table>
| Combination Load Case      | Opens the **Define Load Combinations** dialog, which is used to specify a load case that combines results of analysis performed for different primary load cases.  
**Note:** That this option combines the results of the analysis in the specified manner. It does not analyze the structure for the combined loading. |
| Reference Load Case        | Opens the **Create Primary Load Case** dialog, which is used to create new reference load definition.                                         |
# Ribbon Control Reference

## Loading tab

### Table 314: Loading group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Items</td>
<td>Opens the <strong>Create New Load Items</strong> dialog, which is used to</td>
</tr>
</tbody>
</table>

### Table 315: Define Load Systems group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wind</td>
<td>Opens the <strong>Create New Wind Type Definition</strong> dialog, which is used to add a new wind load definition.</td>
</tr>
<tr>
<td>Direct Analysis</td>
<td>Opens the <strong>Add New Direction Analysis Definition</strong> dialog, which is used to define parameters used in the direct analysis method described in AISC 360-05 Appendix 7.</td>
</tr>
<tr>
<td>Snow</td>
<td>Opens the <strong>Add New Snow Definition</strong> dialog, which is used to define parameters used for generating snow loading on a structure per the ASCE 7-02 code.</td>
</tr>
</tbody>
</table>
Vehicle

Opens the Add New Vehicle Definitions dialog, which is used to define different types of moving loads. Several load cases can be generated by applying these types of loads.

Seismic > <country list>

Opens the Add New Seismic Definitions dialog, which is used to define the parameters for performing a dynamic analysis using the static equivalent approach as outlined in the various seismic codes supported by SPRO.

Pushover

Opens the Add New: Pushover dialog, which is used to define structural parameters for a pushover analysis.

Table 316: Dynamic Specifications group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Forcing Function</td>
<td>Opens the Add New Time History Definitions dialog, which is used to define the Forcing Function of a time varying load.</td>
</tr>
<tr>
<td>Parameter(s)</td>
<td>Opens the Define (Time History) Parameters dialog, which is used to define time step, damping, and arrival times for time history loads.</td>
</tr>
<tr>
<td>Modal Damping</td>
<td>Opens the Modal Damping dialog, which is used to define unique damping ratios for the individual modes used in a dynamic analysis.</td>
</tr>
</tbody>
</table>

Table 317: Display group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load</td>
<td>Select the active load case, load combination, or load envelope from the pop-up dialog.</td>
</tr>
<tr>
<td>View Loading Diagram</td>
<td>Click to toggle the display of the current load case on the structure.</td>
</tr>
</tbody>
</table>

Load & Definition dialog

Used to create and assign load types to the structure.
Opens when the Loading page is selected in the Analytical Modeling workflow.
**Load & Definitions list**

- **Definitions** - contains the options through which you create the DEFINE block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases.
- **Load Cases Details** - contains the dialog boxes through which load cases can be created and loading data added to such cases.
- **Load Envelopes** - Lists all load envelopes contained in the model. Selecting the Load Envelopes and click the Add button to open the Add New Load Envelopes dialog.

**New**
Opens the Create New Definitions/Load Cases/Load Items dialog, which is used to define new Primary Load Cases.

**Add**
Click to add the newly created load under the current load case in the Load dialog box.

**Edit**
Used to modify a previously assigned load component. When clicked, the corresponding load dialog is opened with additional controls for editing the load and structural objects to which the load is applied.

**Delete**
Removes a previously assigned load case or load component.

**Toggle Load**
Select this option to enable what is called the toggle mode. In this mode, when an attribute is selected and assigned using the **Use Cursor to assign** method, the following happens.

- Click on the member or element once - the attribute gets assigned.
Assignment Method

These options are used to assign predefined loads to nodes, members, and elements. First select a predefined load specification from the Loads & Definitions list. Use one of the following assignment methods to apply this load: Assign To Selected Beams, Assign To View, Use Cursor To Assign or Assign To Edit List.

- **Assign To Selected Beams** - To assign a load to selected nodes, members or elements, first select the load from the Loads dialog box. The selected load is highlighted. Then select the nodes, members or elements to which this load is to be assigned. This is done by going to the Select menu, then choosing the appropriate cursor option. Select the desired nodes, members or elements using the cursor. When all desired geometry is selected, click the Assign To Selected Beams radio button, then click the Assign button. Note that the label for this button changes depending on whether nodes, members or elements are selected.

- **Assign To View** - To assign a load to all nodes, members or elements in a view, first select the load from the Loads dialog box. The selected load is highlighted. Select the Assign To View radio button, then click the Assign button. All nodes, members or elements in the structure are assigned this load.

- **Use Cursor To Assign** - To assign a load to nodes, members or elements using the cursor, first select the load from the Loads dialog box. The selected load is highlighted. Select the Use Cursor To Assign radio button, then click the Assign button. The button will appear depressed and the label will change to Assigning. Use the appropriate cursor (from the Select menu) to assign the selected load to the individual nodes, members or elements. Click on the Assign button again to finish.

- **Assign To Edit List** - To assign a load using a typed list of node, member or element numbers, first select the load from the Loads dialog box. The selected load is highlighted. Select the Assign To Edit List radio button, then type the list of node, member or element numbers and click the Assign button.

---

**Create New Definitions / Load Cases / Load Items** dialog box

Used to define new Load Definitions, Primary Load Cases, individual Load Items, and Load Envelopes. The Create New Definitions / Load Cases / Load Items dialog box acts as a master container for many load-related dialogs. Opens when New... is clicked in the Load & Definitions dialog.
Figure 329: Create New Definitions dialog Seismic Parameters tab

This dialog contains the following tabs:
Ribbon Control Reference

Loading tab

Definitions tab

This contains the options through which one creates the “Define” block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases. The command syntax for such cases is explained in TR.31 Definition of Load Systems (on page 2540).

<table>
<thead>
<tr>
<th>Tab name</th>
<th>Dialog Contained</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seismic</td>
<td>Seismic Parameters</td>
</tr>
<tr>
<td></td>
<td>Self Weight</td>
</tr>
<tr>
<td></td>
<td>Joint Weights</td>
</tr>
<tr>
<td></td>
<td>Member Weights</td>
</tr>
<tr>
<td></td>
<td>Element Weights</td>
</tr>
<tr>
<td></td>
<td>Floor Weights</td>
</tr>
<tr>
<td></td>
<td>Reference Load</td>
</tr>
<tr>
<td>Time History</td>
<td>Define Time History</td>
</tr>
<tr>
<td></td>
<td>Define Parameters</td>
</tr>
<tr>
<td>Moving</td>
<td>Define Load</td>
</tr>
<tr>
<td></td>
<td>AASHTO Spec</td>
</tr>
<tr>
<td></td>
<td>File Input</td>
</tr>
<tr>
<td>Wind</td>
<td>Create Wind Type Definition</td>
</tr>
<tr>
<td>Snow</td>
<td>Snow Type Definition</td>
</tr>
<tr>
<td>Pushover</td>
<td>Add New Pushover</td>
</tr>
<tr>
<td></td>
<td>Define Input</td>
</tr>
<tr>
<td></td>
<td>Define Loading Pattern</td>
</tr>
<tr>
<td></td>
<td>Define Spectrum Details</td>
</tr>
<tr>
<td></td>
<td>Define Hinge Property</td>
</tr>
<tr>
<td></td>
<td>Define Solution Control</td>
</tr>
</tbody>
</table>

Load Case tab

Used to generate new load cases (primary and load combination) as well as moving load generations.
### Ribbon Control Reference

#### Loading tab

<table>
<thead>
<tr>
<th>Tab name</th>
<th>Dialog Contained</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary</td>
<td>Create Primary Load Case dialog (on page 3002)</td>
</tr>
<tr>
<td>Moving</td>
<td>Create New Load Generation dialog (on page 3031)</td>
</tr>
<tr>
<td></td>
<td>(Moving) Load Generation Type dialog (on page 2999)</td>
</tr>
<tr>
<td>Combination</td>
<td>Define Load Combinations dialog (on page 3003)</td>
</tr>
<tr>
<td></td>
<td>Create Auto Load Combinations dialog (on page 3033)</td>
</tr>
</tbody>
</table>

#### Load Items tab

Load Items contains the dialog boxes through which loading data can be added to load cases.

This tab contains the **Create New Load Item** dialog (on page 3006).

#### Load Envelopes tab

This tab contains the **Add New Load Envelopes** dialog (on page 3000).

- **Add**: Adds the specified load data to the model.
- **Close**: Closes the dialog without adding any load data.
- **Help**: Opens the STAAD.Pro Help window.

#### (Moving) Load Generation Type dialog

Used to specify direction on plane on the structure to simulate the movement of a predefined vehicle for a moving load definition.

**Note:** If a moving load generation command has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the **Create New (Moving) Load Generation** dialog (on page 3031) to learn more about defining a moving load.

Opens when **Add** is clicked from the **Load & Definition** dialog when **Vehicle Definitions** is selected.

**Note:** Refer to Section 5.31.1 of the Technical Reference Manual for additional information.

- **Type**: Reference a previously defined moving load (e.g., vehicle definition) in the input file. Vehicles definitions are created using the **Create New Moving Load Definitions** dialog (on page 3048).
- **Range**: (Optional) defines section of the structure along global vertical direction to carry moving load. This r value is added and subtracted to the reference vertical coordinate (y1 or z1) in the
global vertical direction to form a range. The moving load will be externally distributed among all members within the vertical range thus generated. r always should be a positive number. In other words, the program always looks for members lying in the range Y1 and Y1+ABS(r) or Z1 and Z1+ABS(r). The default r value is very small, so entering r is recommended.

Initial Position of Load

Load Increment

Global X, Y, and Z coordinates of the initial position of the reference load.

Incremental values of position of load system coordinates (in global axes) to be used for generation of subsequent load cases.

**Note:** The defined vertical direction (e.g., either Y-up or Z-up) is not available to incremental placement of the moving load.

Related Links

- [M. To add vehicles to the load generation](on page 855)

**Add New Define Starting Mass Load** dialog

Used to explicitly define the starting load vector for use with Ritz vector analysis.

The direction selected will extract a single set of Ritz vectors. This is thus best suited when the structural response is predominant in one translation direction. If responses in multiple translational degrees of freedom are predominant, then it is recommended that the program generated starting load vector is used (i.e., do not define a starting mass load with the load dependant Ritz vector eigen solution method for this case).

For each direction selected, enter the number of vectors to generate in that direction.

**Tip:** The number of vectors should typically be between 10 and 35, inclusive. If this number of vectors fails to achieve 90% mass participation in the direction of interest, then it is recommended to use full Ritz vector or Eigen solution instead.

- X Direction Only
- Y Direction Only
- Z Direction Only
- X and Y Directions
- Y and Z Directions
- Z and X Directions

Related Links

- [G.17.3.1 Solution of the Eigenproblem](on page 2363)
- [M. To use starting vectors with load-dependant Ritz vectors](on page 865)

**Add New Load Envelopes** dialog

Used to create load envelopes, which are used to group results set under a single id and can later be used in post processing. If one or more tasks have to be performed for a set of load cases, such as, serviceability checks under steel design for one set of load cases, strength checks under steel design for another set of cases, etc., this feature is convenient.
Note: Load Envelopes are an alternative to using Load Lists. These commands are treated the same by the STAAD engine.

Once load envelopes have been defined, you can select a Load Envelope for reporting results or design in the Result Setup dialog or the appropriate design setup dialog.

Opens when Add is clicked on the Load & Definition dialog with the Load Envelopes entry selected.

**Envelope**
Specify a unique ID number for the load envelope.

**Type**
The envelope can be tagged with optional key words to specify qualitative nature of the load or load combination cases included in the envelop definition. Based on the nature of the load cases in the envelope, you can define appropriate design parameters for each envelope.

For example, for design under wind load condition, most of the design codes allow increase of allowable stresses. Design routine can increase the allowable stress used in interaction equation, when it does the design for the envelope. Another application of this feature can be to specify separate load groups for serviceability check, working stress and limit state checks.

**Available list**
All primary load and load combinations cases are listed here.

**List Operators**

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected load case to the Selected list</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all load cases to the Selected list</td>
</tr>
</tbody>
</table>
Click this button… | to…
---|---
<< | Remove all entries in the Selected list, load combination is placed back in the Available list.
< | Remove the selected entry from the Selected list.

**Selected list**  Loads to be considered in this Load Envelope.

**Related Links**
- *M. To create a load envelope* (on page 877)

## Create Primary Load Case dialog

Used to create new primary load cases.

Opens when the **Primary Load Case** tool is selected from the **Loading Specifications** group on the **Loading** ribbon tab.

### Number

Specify a the load case Number the program recommends, or specify one of our own.

**Note:** If the load case number has been previously used in this input file, a warning message will be displayed.

### Loading Type

Used to associate the load case we are creating with any of the ACI, AISC, or IBC definitions of Dead, Live, Ice, etc. This type of association needs to be done in order to make use the program's facility for automatically generating load combinations in accordance with those codes.

The available load types are the following:

<table>
<thead>
<tr>
<th>Dead</th>
<th>Soil</th>
<th>Accidental</th>
</tr>
</thead>
<tbody>
<tr>
<td>Live</td>
<td>Rain Water / Ice</td>
<td>Flood</td>
</tr>
<tr>
<td>Roof Live</td>
<td>Ponding</td>
<td>Ice</td>
</tr>
<tr>
<td>Wind</td>
<td>Dust</td>
<td>Wind on Ice</td>
</tr>
<tr>
<td>Seismic</td>
<td>Traffic</td>
<td>Crane Hook</td>
</tr>
<tr>
<td>Snow</td>
<td>Temperature</td>
<td>Mass</td>
</tr>
<tr>
<td>Fluids</td>
<td>Imperfection</td>
<td></td>
</tr>
</tbody>
</table>

### Reducible per UBC/IBC

(for Loading Type = Live only) Select this option if the load will be a reducible load per building code.
The UBC 1997, IBC 2000, and IBC 2003 codes permit reduction of floor live loads under certain situations. Please refer to TR.32.4 Area, One-way, and Floor Load Specifications (on page 2664) for details of this feature.

**Title**
Type an optional title which is used to describe the load for identification in modeling and in reports.

**Related Links**
- *M. To create a new primary load case* (on page 822)

**Define Load Combinations** dialog

Used to specify a load case that combines results of analysis performed for different primary load cases.

Opens when the **Combination Load Case** tool is selected from the **Loading Specifications** group on the **Loading** ribbon tab.

**Tip:** To automatically generate multiple combinations, use the **Edit Load Rules for Auto Load Combination Generator** dialog.

For more information about Load Combinations, refer to TR.35 Load Combination Specification (on page 2791).

**Load No.**
Specify an integer value to define a new load case combination. This number is automatically incremented.

**Name**
Specify a *Name* for the load case combination.

**Type**
There are three types of combinations possible:

- **Normal** - A pure algebraic load combination in which the results of all constituent cases will be algebraically added. To accomplish this, select each case individually from the left pane, specify the value by wish we see its results factored, and move it over to the right pane by clicking on the > arrow. Do not check any of the boxes Combine Algebraically, SRSS or ABS.

- **SRSS** - The square root of the sum of the squares method of combination.

  This enables one to achieve a combination such as \( L1 \times f1 + r1 \times \sqrt{(f2*L2^2 + f3*L3^2)} \).

  The sequence of operations will be,

  1. Select the **SRSS** check box. The **Combine Algebraically** check box will become active.
  2. Select \( L1 \). In the box above **Combine Algebraically**, specify a **Factor**, (term \( f1 \) in the equation above), place a check against the **Combine Algebraically** check box, click on the > arrow.
  3. Select \( L2 \). In the box above **Combine Algebraically**, specify a **Factor**, (term \( f2 \) in the equation above), un-check the **Combine Algebraically** check box, click on the > arrow.
4. Select L3. In the box above Combine Algebraically, specify a Factor, (term f3 in the equation above), keep the Combine Algebraically box un-checked, click on the > arrow.

5. In the box called Factor which appears to the right of the expression SRSS, specify the value for r1.

- **ABS** - Algebraic combination of absolute values.

  This enables one to achieve a combination such as f1*ABS(L1) + f2*ABS(L2). The sequence of operations will be,

  1. Switch on the ABS check box.
  2. Select L1. In the box above Combine Algebraically, specify a Factor, (term f1 in the equation above), click on the > arrow.
  3. Select L2. In the box above Combine Algebraically, specify a Factor, (term f2 in the equation above), click on the > arrow.

**Factor**

The load factor coefficient, b, used for the square root value of the sum of the squares.

**Default**

Indicates the default value of the load factor, a (or c, in the case of a SRSS combination), for any load case moved from the Available Load Cases to the Load Combination Definition.

Load factors may be changed from the current default by selecting the entry in the Load Combination Definition.

**SRSS Component**

(Active only when the SRSS type is selected) The SRSS load combination type allows a mixture of algebraic combination with an SRSS combination. Select this option to for the load to include it as a SRSS in lieu algebraic addition.

**Generate Combination**

(Active only when either the SRSS or ABS type is selected) Se this option to generate 1 - 64 load combinations for each combination of degrees of freedom at member ends to create load envelopes for positive and negative analysis results for SRSS and Absolute load combinations.

**Available Load Cases list**

Lists all of the previously created Primary Load Cases or Load Combinations in the current input file.

**Tip:** Double clicking any load case or combination will add it to the Load Combination Definition list.

**Load Combination Definition list**

Entries here represent the Primary Load Case, load factors, and SRSS component (if any) which will make up the Load Combination.

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected load case or combination to the Load Combination Definition list</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all load cases and combinations to the Load Combination Definition list</td>
</tr>
</tbody>
</table>
Click this button… | to…
---|---
<< | Remove all entries in the Load Combination Definition list.
< | Remove the selected entry from the Load Combination Definition list.

**Example**

Thus, for example, if you want to create a combination case called 101, which combines the absolute values of the results of cases 7 and 15, factored by 1.5 and 1.64 respectively, the command would be:

```
LOAD COMBINATION ABS 101
7 1.5 15 1.64
```

From the Create New Load Combination dialog:

1. Select the **ABS** option.
2. Select load case 7 in the Available Load Case list.
3. Specify a **Default** factor of 1.5 and move it to the right pane by clicking on the **>** button.
4. Repeat this process for case 15 with a factor of 1.64.
5. Click **Add**.

The table shown below illustrates the Load Combination logic for Algebraic and SRSS types assuming that the Primary Load Cases are L1, L2, and L3.

<table>
<thead>
<tr>
<th>Primary Load Case</th>
<th>Factor</th>
<th>Combin e Algebraically?</th>
<th>SRSS?</th>
<th>SRSS Factor</th>
<th>Resulting Combination Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>L1</td>
<td>0.75</td>
<td>Yes</td>
<td>No</td>
<td>n/a</td>
<td>0.75(L1) + 0.75(L2) + 1.33(L3)</td>
</tr>
<tr>
<td>L2</td>
<td>0.73</td>
<td>Yes</td>
<td>No</td>
<td>n/a</td>
<td></td>
</tr>
<tr>
<td>L3</td>
<td>1.33</td>
<td>Yes</td>
<td>No</td>
<td>n/a</td>
<td></td>
</tr>
<tr>
<td>L1</td>
<td>1.0</td>
<td>No</td>
<td>Yes</td>
<td>1.2</td>
<td>1.2√[L1]^2 + 0.4[L2]^2 + 0.4[L3]^2</td>
</tr>
<tr>
<td>L2</td>
<td>0.4</td>
<td>No</td>
<td>Yes</td>
<td>1.2</td>
<td></td>
</tr>
<tr>
<td>L3</td>
<td>0.4</td>
<td>No</td>
<td>Yes</td>
<td>1.2</td>
<td></td>
</tr>
</tbody>
</table>

**Related Links**

- [M. To define a new load combination](on page 866)
**Add New Reference Load Definitions** dialog

Used to add a reference load case to the input file.

Opens when the **Reference Load Case** tool is selected in the **Loading Specifications** group on the **Loading** ribbon tab.

---

**Note**: See [GS. Load Types in STAAD.Pro](on page 47) for additional information on when different load types are used in STAAD.Pro.

The parameters in this dialog are analogous to the **Create Primary Load Case** dialog (on page 3002).

**Note**: In STAAD.Pro V8i (SELECTseries 3) and higher, the Loading Type Mass can be used to generate a single mass model for re-use in dynamic loads. Refer to [TR.31.6 Defining Reference Load Types](on page 2642) for additional details.

---

**Add New Load Items** dialog

Used to define and assign loads to the structure.

Opens when the **Load Items** tool is selected in the **Loading Specifications** group on the **Loading** ribbon tab.

---

Load Item types are grouped into tabs on the right side of the dialog, with parameters for each type in the main dialog.
Note: If the current active load case is not a Primary Load, then many of the tabs will display an error message and the controls will be inactive.

**Load Specifications list**
Lists the categories into which load specifications are grouped. Selecting a load specification type causes the appropriate dialog controls to be displayed to the right of the list.

**Add**
Adds the load case on the current tab to the input file. The new load appears in the **Load & Definition** dialog under the currently selected load case.

**Close**
Closes the dialog. Any load data entered without clicking the Add button will be lost.

**Help**
Opens the STAAD.Pro help window.

**Selfweight tab**
Used to apply the selfweight of the structure as a load. Selfweight of all active components of the structure are calculated and applied as a uniformly distributed load.
**Direction**  Specify the direction in which the selfweight load is to be applied by clicking on the X, Y, or Z buttons.

**Factor**  Specify the factor with which the calculated selfweights are to be multiplied. A negative value indicates that the load is applied along the negative direction of the chosen axis.

**Related Links**
- *M. To add selfweight load* (on page 823)

---

**Nodal Load tab**

Used to apply nodal loads.

The following options are available under this tab:

**Node**

- **Fx/Fx’, Fy/Fy’, Fz/Fz’**  Type values of the forces in corresponding global directions or inclined axis (’) directions when the Inclined Load? option is checked. A negative sign (-) may be used.
- **Mx/Mx’, My/My’, Mz/Mz’**  Type values of the moments about the corresponding global directions or inclined axis (’) directions when the Inclined Load? option is checked. A negative sign (-) may be used.

**Inclined Load?**  Check this option to specify the load along an inclined coordinate system. An inclined coordinate system is defined by an X’ axis that pass from the loaded joint through a reference point. That reference point can be defined either a reference node, a set of absolute (global) coordinates, or a set of relative distances from the loaded joint.

**Inclination**  Select the option used to define the inclined coordinate system.

- **Reference Node** - select an existing node number from the model. The global coordinates of that node are displayed as read-only.
- **Absolute** - type the global coordinates of an arbitrary point to use as the reference.
- **Relative** - type relative distances from the loaded joint to the reference point. These distances are measured along the global axes.
The reference point—whether it is specified using absolute coordinates, relative distances, or a reference joint—defines the direction of the $x'$ axes from the loaded node to the reference point. The direction of the $y'$ axes is then defined as the direction perpendicular to $x'$ that lies in the plane of $x'$ and $y$. In the special case of when $x'$ is in the direction of $y$, $y'$ is taken to be the direction of $z$. The direction of $z'$ is then defined as the direction perpendicular to the plane $x'$ and $y'$ and follows the right-hand rule as for all other axes systems used in STAAD.Pro.

Support Displacement

Used to specify a support displacement. Multiple support displacements may be used for a single load case, but each support value and direction must be entered independently.

- **Displacement** Specify the value of the displacement. A negative sign (-) may be used.
- **Direction** Select the direction of the displacement as $F_x$, $F_y$, or $F_z$ (translational), or $M_x$, $M_y$, or $M_z$ (rotational).

Related Links

- TR.31.2.14 IBC 2015 Seismic Load Definition
  - M. To add a nodal load (on page 824)
  - TR.31.2.14 IBC 2015 Seismic Load Definition (on page 2600)
  - G.15.1 Joint Loads (on page 2337)
  - M. To add a nodal load (on page 824)
  - G.15.7 Support Displacement Loads (on page 2342)
  - M. To add a support displacement (on page 825)

Member Load tab

Used to apply loads on the span of frame members or physical members (select the appropriate parent tab).

These following options are available:
Uniform Force / Moment

Used to specify a Uniformly Distributed Force or Uniformly Distributed Moment along a member.

- **W1** Value of the load in currently selected units.
- **d1, d2** Distance of starting and ending points of the load from the starting node (Node A) of the member. If these values are zero, then the load is applied over the entire length of the member.
- **d3** Perpendicular distance from the member's shear center to the plane of loading.

**Direction** Click on the appropriate radio button to specify the direction of the load. \( X, Y, Z \) indicate the direction in local coordinates; \( GX, GY, GZ \) indicate the loads in global coordinates; \( PX, PY, PZ \) indicate the loads along the projected length of the member in the corresponding global direction. However, \( d1, d2 \) and \( d3 \) are still measured along the length of the member and not along the projected length.

Concentrated Force / Moment

Used to specify a concentrated force or moment load on a member.

- **P** Value of the load in currently selected units.
- **d1** Distance of the load from the starting node (Node A) of the member. If this value is not specified, then the load is applied at the midpoint of the member.
- **d2** Perpendicular distance from the member's shear center to the plane of loading.

**Direction** Click on the appropriate radio button to specify the direction of the load. \( X, Y, Z \) indicate the direction in local coordinates; \( GX, GY, GZ \) indicate the loads in global coordinates; \( PX, PY, PZ \) indicate the loads along the projected length of the member in the corresponding global direction. However, \( d1 \) and \( d2 \) are still measured along the length of the member and not along the projected length.

Linear Varying

Used to specify a linearly varying load on a member. The load is applied over the entire length of the member.

- **W1, W2** For a linearly increasing or decreasing load, enter the values of force \( W1 \) at the start of the beam and \( W2 \) at the end of the beam in currently selected units.
- **W3** For a triangular load distribution, enter the value \( W3 \) of the force in the middle of the beam.

**Direction** Select the direction of the force in local coordinates from the radio buttons \( X, Y \) or \( Z \).

Trapezoidal

Used to specify a trapezoidal load on a member.

- **W1, W2** Starting and ending load values in currently selected units.
- **d1, d2** Distance of starting and ending points of the load from the starting node (Node A) of the member. If these values are zero, the load is applied over the entire length of the member.

**Direction** Click on the appropriate radio button to specify the direction of the load. \( X, Y, Z \) indicate the direction in local coordinates; \( GX, GY, GZ \) indicate the loads in global coordinates; \( PX, PY, PZ \) indicate the loads along the projected length of the member in the corresponding global direction.
However, d1 and d2 are still measured along the length of the member and not along the projected length.

Hydrostatic

Used to specify loads due to hydrostatic pressure on one or more adjacent beams. The Hydrostatic load is converted to Trapezoidal loads on the beams. The load is applied over the entire length of the members.

**W1, W2**

Type the value of the load at the minimum and maximum global axis in currently selected units. For example, to model a retaining wall with soil pressure, **W1** is the force at the bottom of the wall and **W2** is the force at the top of the wall.

**Direction**

Select the direction in which the force is applied by clicking on the appropriate radio button. **X**, **Y**, **Z** indicate the direction in local coordinates; **GX**, **GY**, **GZ** indicate the loads in global coordinates; **PX**, **PY**, **PZ** indicate the loads along the projected length of the member in the corresponding global direction.

**Interpolate along Global Axis**

Specify the global axis along which the load would vary from **W1** to **W2**. For example, the load would vary along the Y axis on a vertical retaining wall.

**Select Member**

Opens a **Selected Item(s)** dialog, which is used to generate a list of selected members. The Create Loads dialog is dismissed until all loads are added using the Beams Cursor tool in the View Window. Once the **Done** button is clicked, the Create Loads dialog re-opens with the member table populated. Unlike other load definition options, we must select members for this option to become active.

Pre/Post Stress

Used to apply Poststress and Prestress loads on members.

**Type**

Select either the **Prestress** or **Poststress** radio button.

**Load**

Type the **Force** as a positive value in current units.

**Eccentricity Distances**

If applicable, specify the Eccentricity of the cable with respect to the center of gravity of the cross section at the **Start**, **Middle** and **End** of the member.

**Note:** Please refer to **TR.32.5 Prestress Load Specification** (on page 2678) for more detailed information on these options.

Fixed End

If a load acts at an intermediate point along the span of a frame member, there are two ways of assigning the load. a) It could be specified using the member load option. In this case, the magnitude and position along span are specified. b) It could be converted to fixed end actions and defined as a FIXED END load as shown below.

**Start / End Node**

Type values for FX, FY, FZ, MX, MY and MZ (axial, shear-y, shear-z, torsion, moment-y, and moment-z) in the local coordinates for the Start node and/or the End node.

**Related Links**

- **G.15.2 Member Load** (on page 2337)
- **M. To add a concentrated force or moment on members** (on page 826)
- **M. To add a uniform load to members** (on page 826)
Area Load tab

Used to apply area (panel) load which will be distributed on surrounding beams based on a one way distribution.

**Pressure**  This load is a one-way distributed pressure load on members that circumscribe a panel. Enter the value of the area load (force per unit area) in the current units. This load always acts along the positive local y-axis on the two longest members in each panel.

**Direction**  Select the direction in which the force is applied. **Local Z** indicates the direction in local coordinates, perpendicular to the panel; **GX, GY, GZ** indicate the loads in global coordinates.

**Related Links**
- G.15.3 Area, One-way, and Floor Loads (on page 2338)
- M. To add an area load (on page 830)

Floor Load tab

Used to apply a panel load which will be distributed on surrounding beams based on a two way distribution.

This load is a two-way distributed pressure load on members that circumscribe a panel.

| Load | Floor load value (force per unit area) in the current units. This load will act parallel to the global vertical axis. |
| X RANGE / Z RANGE / Y RANGE options | Select one of these options to specify a floor load by range of global coordinates |
| Group option | Select this option to apply a floor load on a previously added member group or composite deck definition. |
| Pressure | Specify the pressure of the floor load in the current force per square length units. |
| Direction | The floor load may be considered as acting perpendicular to the plane of the panel on which it is defined. This is the normal static load condition. However, for the purposes of generating the mass matrix for dynamic analysis, it may also be considered as weights lumped at the nodes of its panels and acting (vibrating) parallel to the plane of the panel. Hence, the user has the choice of specifying it in one of the 3 global directions. |

**Note:** Refer TR.32.4.3 Floor Load Specification (on page 2672) for more information.

| Define X Range / Y | Specify the location of the floor using the *Define X Range* option. The load will be calculated for all members lying between this range. |
Range / Define Z Range
Corner points of the area on which the floor load acts, in global X and Z directions. If omitted, the floor load is assumed to be acting for all members located within the Y range specified.

One Way Distribution
Select this option to use one way distribution to get a one way type distribution of the pressure. In such cases the program finds out the shorter side of the panel. It then divides the load in between the long direction beams. No load is generated by this option if the panel is square in shape.

To specify a floor load with the Z-axis vertical, select the Floor With Z Range menu option. The data items are the same as the Floor load with Y axis vertical except that the Y and Z axes are interchanged.

Towards
(One Way Distribution) Used to define the direction in which the load is spanning by selecting a member in the loaded zone onto which the load is to be directed. The default is for the load to resolve toward the longer span of supporting beams framing the Floor load. However, you can select a beam number to direct the load to that beam.

Panel Information Output group
Includes the floor panel information in the output file (.ANL).

Print Options (output file):
- **Print** - The total number of panels identified, total area of all panels and total load generated will be printed in the output file.
- **Panel with member list** - This option is used to print panel member numbers on which loads are generated in addition to the Print option output.
- **Panel with member load** - This option is used to print panel loading information in addition to the Print with member list option output.

Print to external text file
This option will output the panel information in an external text file named *(filename)_FLD.TXT*.

Print Options (external text file):
- **Panel with member list** - This option is used to print panel member numbers on which loads are generated. Additionally, the total number of panels identified, total area of all panels and total load generated will be printed in the text file.
- **Panel with member load** - This option is used to print panel loading information in addition to the Print with member list option output.

Related Links
- **G.15.3 Area, One-way, and Floor Loads** (on page 2338)
- **M. To add a floor load or one-way load** (on page 831)

Plate Loads tab
Used to apply loads to elements.

Pressure on Full Plate
Used to define a pressure load that acts on the full surface of an element. (To define a pressure load that acts on a small part of an element, see the Partial Plate Pressure tab that is described in the coming pages).
Load  
 Direction  

W1 is the variable using which the pressure value is defined, in pressure units.

The load may be applied along the local Z axis, or along one of the global X, Y or Z axes (GX, GY, GZ).

**Tip:** Loads in the Local X or Local Y directions can be used to represent in-plane friction loads.

### Concentrated Load

Use this option to define a concentrated load that acts at a specific point within the boundary of an element. If a load acts at a node point of an element, it is advisable to apply it using the Nodal Load option described in earlier pages.

**Load**  
**Force** is the magnitude of the load.

**Direction**  
The load may be applied along the local Z axis, or along one of the global X, Y or Z axes (GX, GY, GZ).

### Partial Plate Pressure Load

Used to specify a uniform pressure load on the entire element or on a user specified portion of the element.

**Load**  
The element pressure (force per unit area) or concentrated load (force unit). For concentrated load, the values of X2 and Y2 must be omitted, while X1 and Y1 must be specified.

**X1, Y1, X2, Y2**  
For element pressure (force per unit area), these values represent the coordinates of the rectangular boundary on which the pressure is applied. If X1, Y1, X2 and Y2 are all zero, the pressure is applied over the entire element. If X1 and Y1 are specified but X2 and Y2 are omitted, then W1 is treated as a concentrated load.

**Direction**  
**GX, GY, GZ** represent the global X, Y, and Z directions along which the pressure may be applied. **Local Z** indicates that the pressure is applied normal to the element in the local Z direction.

**Note:** For more information, please refer to [TR.32.3 Element Load Specifications](on page 2657).

### Trapezoidal

Used to specify a trapezoidally varying pressure load on a plate. The load is applied over the entire element in the local Z direction, varying along the positive local X or Y direction.

**Direction of Pressure**  
**GX, GY and GZ** represent the global X, Y, and Z directions along which the pressure may be applied. **Local Z** indicates that the pressure is applied normal to the element in the local Z direction.

**Variation along element**  
Define the direction in which the pressure varies as either the local X or Y direction or choose the joint option which is discussed next.

The Joint option is used to apply different values of pressure at different nodes of the plate element. When checked, the dialog updates to allow a different pressure value for each of the four plate corner nodes.

### Hydrostatic

Used to model loads due to hydrostatic pressure on one or more adjacent elements. The Hydrostatic load is converted to Trapezoidal loads on the elements. The load is applied over the entire area of the element.
**Force**

Enter value of the load at the minimum and maximum global axis in current units. For example, to model a retaining wall with soil pressure, \( W_1 \) is the force at the bottom of the wall and \( W_2 \) is the force at the top of the wall.

**Interpolate along Global Axis**

Specify the global axis (\( X \), \( Y \), or \( Z \)) along which the load should vary from \( W_1 \) to \( W_2 \). For example, the load would vary along the \( Y \) axis on a vertical retaining wall.

**Direction of pressure**

Specify the direction of design pressure as *Local Z* axis or global axes (\( GX \), \( GY \) or \( GZ \)) and click on *Add*. This will assign the linearly varying hydrostatic load on all the selected elements.

**Select Plate(s)**

Opens a *Selected Item(s)* dialog, which is used to generate a list of selected plates. The *Create Load Items* dialog is dismissed until all loads are added using the Plates Cursor tool in the View Window. Once *Done* is clicked, the *Create Load Items* dialog re-opens with the Plate table populated. Unlike other load definition options, we must select plates for this option to become active.

### Element Joint Load

To specify a varying pressure at each joint on a plate, select the *Element Joint Load* option as shown below.

**Joint Load Data**

Choose Three Noded Facet / Four Noded Facet, depending on whether the plate element is 3 noded or 4 noded.

**Pressure Table**

Specify the corner node number and corresponding pressure load, in the current units (displayed).

**Direction**

The load may be applied along the local Z axis, or along one of the global X, Y or Z axes (\( GX \), \( GY \), \( GZ \)).

### Related Links

- [G.15.8 Loading on Elements](on page 2342)
- [M. To add pressure load on a plate](on page 829)

### Surface Loads tab

Used to apply uniform pressure on the faces of solid elements.

**Note:** Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

**Note:** For more information, please refer to [TR.32.3.4 Surface Loads Specification](on page 2663).

### Concentrated Load

**Load**  

*Force* is the magnitude of the load.

**Direction**

The load may be applied along the local Z axis, or along one of the global X, Y or Z axes (\( GX \), \( GY \), \( GZ \)).

### Pressure on Full Surface

**Load**

\( W_1 \) is the variable using which the pressure value is defined, in pressure units.

**Direction**

The load may be applied along the local Z axis, or along one of the global X, Y or Z axes (\( GX \), \( GY \), \( GZ \)).
Partial Surface Pressure Load
To apply a partial pressure load on a surface, provide the magnitude of the pressure load in the $W_1$ edit box. $X_1$ $Y_1$ and $X_2$ $Y_2$ represent the coordinates of the extremities of the loaded area. The coordinates are with respect to a local coordinate system with the origin at the first node of the surface element. Using the Direction option, specify the direction of application of the load which can be either in the local $Z$ direction or in any of the global directions ($GX$, $GY$ or $GZ$).

Load
The surface pressure (force per unit area) or concentrated load (force unit). For concentrated load, the values of $X_2$ and $Y_2$ must be omitted, while $X_1$ and $Y_1$ must be specified.

$X_1$, $Y_1$, $X_2$, $Y_2$
For surface pressure (force per unit area), these values represent the coordinates of the rectangular boundary on which the pressure is applied. If $X_1$, $Y_1$, $X_2$ and $Y_2$ are all zero, the pressure is applied over the entire element. If $X_1$ and $Y_1$ are specified but $X_2$ and $Y_2$ are omitted, then $W_1$ is treated as a concentrated load.

Direction
$GX$, $GY$, $GZ$ represent the global $X$, $Y$, and $Z$ directions along which the pressure may be applied. $Local$ $Z$ indicates that the pressure is applied normal to the surface in the local $Z$ direction.

Partial Surface Trapezoidal Load
Used to load a portion of a surface with a non-uniform pressure.

Direction of Pressure
$GX$, $GY$ and $GZ$ represent the global $X$, $Y$, and $Z$ directions along which the pressure may be applied. $Local$ $Z$ indicates that the pressure is applied normal to the element in the local $Z$ direction.

$X_1$, $Y_1$, $X_2$, $Y_2$
For element pressure (force per unit area), these values represent the coordinates of the rectangular boundary on which the pressure is applied. If $X_1$, $Y_1$, $X_2$ and $Y_2$ are all zero, the pressure is applied over the entire surface. If $X_1$ and $Y_1$ are specified but $X_2$ and $Y_2$ are omitted, then $W_1$ is treated as a concentrated load.

Variation along element
The varying pressure values at the four corners of the loaded area.

Enter the pressure intensity $F_1$ at the lowest local coordinate location (start) and the intensity $F_2$ at the highest local coordinate location (End). $Start$ and $End$ are defined based on the positive direction of the local $X$-axis or local $Y$-axis.

Solid Loads tab
Used to apply uniform pressure and uniformly varying pressure on the faces of the solid element.

Face Number
Select the face on which we want to apply the load.

Node 1…. Node 4
Specify the pressure at each node on the selected face of the solid element. For example, if the pressure is uniform, the same value ought to be entered in all four columns.

Direction
Specify the direction of design pressure as $Local$ $Z$ axis or global axes ($GX$, $GY$ or $GZ$) and click on Add. By doing this, the solid pressure load is added to that particular load case.

Temperature Loads tab
Used to apply Strain and Temperature loads on members.
**Temperature**

Used to specify a temperature load

**Temperature Change for Axial Elongation**

The change in temperature that will cause axial elongation in members or uniform volume expansion in elements.

**Temperature Differential from Top to Bottom**

Temperature differential of the member or element (T top surface – T bottom surface).

**Note:** The top surface is in the positive local Y direction for members, and the positive local Z direction for elements. If the **Temperature Differential from Top to Bottom** is omitted, bending is not considered.

**Temperature Differential from side to side (Local Z)**

Accounted for when the temperature difference is not the same across the sides of the member from left to right.

**Strain**

Used to specify a strain load.

**Initial Axial Elongation or Shrinkage**

Enter the initial elongation or shrinkage in the member in current units. A positive value indicates elongation, a negative value indicates shrinkage.

**Seismic Loads tab**

Used to apply previously defined UBC loads on the structure.

**Note:** If a seismic load definition has not yet been added to the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the **Seismic Parameters dialog** (on page 3044) to learn more about defining a seismic load.

**Direction group**

Specify the global direction in which the seismic load is to be generated by selecting the **X Direction**, **Y Direction**, or **Z Direction** radio button.

**Factor**

Specify a multiplying factor, if applicable. The default factor is 1.0.

**Multiplying factor for Accidental Torsion Moment**

Set this option to include the accidental torsion load per the UBC, IBC, 1893, etc. code. Type the **Factor** in the associated field (may be a negative value).

**Multiplying factor for Natural Torsion Moment**

Set this option to include the torsion arising due to static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm. Type the **Factor** in the associated field (must be greater than or equal to zero).

**Time History tab**

Used to apply previously defined time history loads on the structure.

**Note:** If a time history load has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the **Add New Time History Definition dialog** (on page 3056) or **Define Parameters dialog** (on page 3058) to learn more about defining a time history load.
Loading tab

**Loading Type**
Select **Time Load** to apply the Time History load to joints in the structure or **Ground Motion** to apply the load at the structure’s base.

**Arrival Time**
Select a previously defined Arrival Time to define the time at which the load begins to act.

**Direction**
Select the global direction in which to apply the Time History load.

**Defined Types**
Select a previously defined Type number.

**Force Amplitude Factor**
Specify a factor to multiply the values of force or acceleration which were input while defining the Time History loading.

**Related Links**
- *G.17.3.5 Response Time History* (on page 2373)
- *M. To add a time history load* (on page 861)

Wind Load tab

Used to apply previously created wind load types on the structure through the means of a load case.

**Note:** If a wind load has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the *Add New Wind Type Definition* dialog (on page 3038) to learn more about defining a wind load.

Wind Load tab

Used to add static wind forces per ASCE 7 and other codes.

**Select Type**
Choose a previously defined wind load type from the drop down list.

**Exposed Surface and Direction**
Specify the global direction in which the wind load is to be generated by clicking the X, Z, -X, or -Z option. When wind is generated in X direction, the wind load is applied on the near side and when -X is chosen the load is applied on the far side as explained in the following figures. Generation in Z or -Z is similar.

```
<table>
<thead>
<tr>
<th>Wind in X</th>
<th>Wind in -X</th>
</tr>
</thead>
</table>
```

*Figure 330: Wind in positive and negative directions*
Table 319: Wind direction explanation

<table>
<thead>
<tr>
<th>If the direction is set to…</th>
<th>and $f$ is…</th>
<th>Then the load is applied to surfaces facing…</th>
<th>and the load will be directed toward…</th>
<th>This can be though of as…</th>
</tr>
</thead>
<tbody>
<tr>
<td>X or Z</td>
<td>Positive</td>
<td>Negative X or Z</td>
<td>Positive X or Z</td>
<td>Windward load for wind in the positive X or Z direction</td>
</tr>
<tr>
<td></td>
<td>Negative</td>
<td>Positive X or Z</td>
<td>Negative X or Z</td>
<td>Windward load for wind in the negative X or Z direction</td>
</tr>
<tr>
<td>-X or -Z</td>
<td>Positive</td>
<td>Positive X or Z</td>
<td>Positive X or Z</td>
<td>Leeward load for wind in the positive X or Z direction</td>
</tr>
<tr>
<td></td>
<td>Negative</td>
<td>Negative X or Z</td>
<td>Negative X or Z</td>
<td>Leeward load for wind in the negative X or Z direction</td>
</tr>
</tbody>
</table>

Factor
Specify the factor by which the calculated wind loads will be multiplied.

Open Structure
Select this option to generate loads on open structures like highway signs or transmission towers. By default, the load generation is based on the assumption that the region between members is covered by panels (i.e., cladding). Setting this option will apply load to only the members (i.e., no cladding).

Tip: The term used here is not the same as the ASCE 7 definition of "open structure." Please use the ASCE 7 Wind Load dialog box (on page 3039) to generate describe the enclosure classification per this code.

SNiP Parameters
The following parameters are used for wind loads generated per the Russian code (SNiP).

Apply Wind Load at the Corner
Select this option for wind loads to be applied at 45°.

Note: This parameter is only applicable for rectangular buildings.

Select Configuration
The building configuration as defined in SNiP 2.01.07–85 "Loads and Actions":

0. Prismatic building structure - Rectangular building- Outstanding architectural details on the left façade.
1. Prismatic building structure - Rectangular building - Both side façades are smooth
2. Prismatic building structure - Rectangular building - Outstanding architectural details on the right façade
3. Prismatic building structure - Rectangular building - Outstanding architectural details on both side façades
4. Prismatic building structure - Rectangular building - Triangular building
5. Prismatic building structure - Rectangular building - Rhombic building
6. Prismatic building structure - Rectangular building - Number of vertices of polygonal building, not more than 12
7. Prismatic building structure - Less than 3 — rectangular building
8. Prismatic building structure - 3 — triangular
9. Prismatic building structure - 4 — rhombic
10. Prismatic building structure - More than 4 — polygonal
11. Framed RC structure
12. Lattice steel structure

Reynolds Number (NU) Specify the wind pressure correlation coefficient. If parameter is omitted or is exactly 1, a computed value is used instead. For rectangular buildings, the correlation coefficient is always calculated automatically, thus any specified value will be ignored.

Note: The first load case which has a Russian Wind Load command added to it will consider all other loads defined in it as the masses to be considered for calculating the dynamic effect which is required by this command.

Wind Load - Dynamic tab
Used to specify dynamic wind forces per SP 20.13330.2016.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wind Definition</td>
<td>Choose an SP 20.13330.2016 wind load definition from the drop down list.</td>
</tr>
<tr>
<td>Width of Building along wind dir</td>
<td>Effective length of the structure parallel to wind direction.</td>
</tr>
<tr>
<td>Wid of Building across wind dir</td>
<td>Effective projection of the structure facing the wind direction.</td>
</tr>
<tr>
<td>Factor</td>
<td>Correction factor along specified direction.</td>
</tr>
<tr>
<td>Internal</td>
<td>Set this option to indicate an internal, or suction, wind force on the surfaces (i.e., the load direction is reversed).</td>
</tr>
<tr>
<td>Allow all Mode Shapes</td>
<td>Check this option to compute dynamic wind load vector for all mode shapes extracted from Modal Analysis provided it is within code stipulated frequency limit. When this check box is clear, the calculation takes only the first mode shape.</td>
</tr>
<tr>
<td>Static Wind Case</td>
<td>Select a previously defined primary load cases in the model.</td>
</tr>
<tr>
<td>Wind Direction</td>
<td>Select the plan direction to apply the wind force.</td>
</tr>
</tbody>
</table>
Snow Load tab

Used to generate snow loading on a structure in accordance with the provisions of the ASCE-7-02 code.

The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

**Note:** If a snow load has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the Add New Snow Definition dialog (on page 3043) to learn more about defining a snow load.

**Floor Group**
Select the floor group on which the snow load is to be applied.

**Condition**
Specify whether the load is "Balanced" or "Unbalanced." These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.

**Define Snow Type**
Select the snow load type number. This is the number specified against the Type No. field while creating the snow load definition in Part I described earlier in this section.

**Roof Type**
Specify the roof type from the available choices:
- Default (if the roof type is not Mono, Hipped or Gable, it is referred to as Default)
- Mono (see mono-sloped roof shown in figure 6-6 of the code)
- Hipped (see figures 6-3 and 6-6 of the code)
- Gable (see figures 6-3 and 6-6 of the code)

**Roof Obstruction**
Specify whether the roof is "obstructed" or "unobstructed." This also is a term described in section Table 7-2 of ASCE 7-02.

**Roof Slope Factor**
For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.

**Related Links**
- G.16.4 Snow Load (on page 2345)
- M. To add an ASCE 7-02 snow load (on page 851)

Response Spectra tab

Used to apply response spectrum loads on the structure.

**Note:** The dialog updates dynamically to reflect the Code selection.
Common Parameters for all Code options

Combination Method

The various methods available in STAAD for combining the contribution from the individual modes is listed under this heading. The details of these methods are explained in Section 5.32.10.1 of the Technical Reference manual.

Note: Not all methods are available for all code options.

- **SRSS** is the square root of summation of squares method.
- **ABS** is the absolute sum method. This method is very conservative and represents a worst case combination.
- **CQC** is the complete quadratic combination method. This method is recommended for closely spaced modes instead of SRSS.
- **ASCE** is the NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenbluheth) is used for close mode interaction of the damped periodic portion of the modes. This method should only be used for a general response spectrum.
- **TEN** is the Ten Percent Method of combining closely spaced modes as per the NRC guideline 1.92 (1976).
- **CSM** is the closely spaced modes method. The peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined by Absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method.
- **GRP** is the closely spaced modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined using the absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Save

Select this option to generate a file (with .acc extension) containing the joint accelerations in g’s and radians/sec^2

Period vs. Acceleration table

Provide the values of period (seconds) and corresponding acceleration (current length units/sec^2) or displacement (current length unit). Spectrum pairs should be provided in ascending value of period. As we provide the curve points, the program displays the curve at the bottom of the dialog box.

Spectrum Type

Choose whether the response spectrum curve will be input as Period vs. Acceleration or Period vs. Displacement (Custom and IS 1394 only).

Interpolation Type

From the spectrum data that are provided under the Define Spectrum Pairs tab of the dialog box shown above, STAAD fetches the spectral value for the actual modes of the structure using one of two interpolation methods – **Linear** and **Logarithmic**. Linear interpolation is the default method.

Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

Damping Type

- **Damping** - This is to be used for specifying a single modal damping ratio which will be applied to all modes. The default value is 0.05.
• **CDAMP** - Select this option to use Composite Modal Damping. This evaluates the damping from that defined in the material or constant definitions. A Damping ratio is defined by the material definition or in the **Material Constants - Damping Ratios** dialog. If there is no damping information entered in the material or constant definitions, the behavior is the same as **MDAMP**.

• **MDAMP** - Selection this option to use Modal Damping, which is used for individual damping ratios for each mode. Individual mode damping ratios are defined using the [Modal Damping dialog](on page 3059).

**Scale**

Linear scale factor by which the spectra data will be multiplied. Usually to factor g’s to length/sec\(^2\) units. This input is the appropriate value of acceleration due to gravity in the current unit system.

You may choose to provide the spectral acceleration or displacement data as a set of un-normalized values or as a set of normalized values. For normalized values, the normalization factor is specified through the means of the Scale factor. For example, if the curve is input in terms of “g” - the acceleration due to gravity - and the current length unit is feet, the **Scale** would be 32.2. For un-normalized values, the scale factor is provided as 1.0, which also happens to be the default. The spectra data will be multiplied by the scale factor during the analysis. For more information, please refer to section 5.32.10.1 of the Technical Reference manual.

**Missing Mass**

Select this option to apply the Missing mass correction. The static effect of the masses not represented in the modes is included. For more information, please refer to section 5.32.10.1 of the Technical Reference manual. If this option is selected on any spectrum case it will be used for all spectrum cases.

**ZPA**

Zero Period Acceleration: It is used only with the missing mass option. If no value is entered or a zero value is entered, the default considered by the program is 33 Hz. If an acceleration is entered corresponding to the Missing mass mode, then the ZPA value is ignored. If no acceleration value is entered for the missing mass mode, then spectral acceleration corresponding to the ZPA frequency is used.

**Direction**

Specify the global direction(s) in which the spectrum is to be applied. The response spectrum may be applied in one or more directions simultaneously. Directions not provided will default to zero.

**Use Torsion**

(IS 1893, IBC 2012, and NRC 2005 only)

- **Dynamic Eccentricity (DEC)**
  Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity). If not specified (or zero), a value of 1.0 is assumed.

- **Accidental Eccentricity (ECC)**
  Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion. If not specified (or zero), a value of 0.05 (5% of floor plan dimension perpendicular to force at a given level) is used.

**Signed Response Spectrum Results Options**

Two method are available for added mathematical signs to the spectrum response output:

- **Dominant Mode No.** - Select this option and (optionally) specify a mode number to define as a dominant mode. The sign (sense) of this mode will be applied to other modes.
Signed - Select this option to create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.

Individual Modal Response Load Case Generation

Select this option to have the program automatically generate primary load cases from the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. A load case is generated for each of first number of modes specified, starting with the specified load case number.

Note: The Individual Modal Response case generation is not available for SNiP II code response spectra.

Custom

Generate Spectrum

Opens the Spectrum Parameters dialog (on page 3027), which is used to generate a response spectrum curve per the International Building Code.

Note: This is the response spectrum type that is explained in Section TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2688).

IS 1893

Use Torsion (IS1893)

Select this option to consider the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity during analysis. If this option is used for any one spectrum case it will be used for all spectrum cases.

Soil Type

Check this box to get a pull down menu & choose the soil type for the site (hard, medium or soft). Depending on the type of soil & time period, average response acceleration coefficient Sa/g is calculated.

Note: This is the response spectrum type that is explained in Section TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2721).

EURO (EC8) -1994

Load Type

Select either a Elastic or Design response spectrum for the loading type.

Design Ground Acceleration

Specify a design ground acceleration expressed in terms of acceleration due to gravity (g). For most of the application of Eurocode 8, the hazard is described in terms of a single parameter (i.e., the value of effective peak ground acceleration in rock or firm soil). This acceleration is termed as the design ground acceleration.

Behaviour Factor

Specify the value used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

Subsoil Class

Used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of three kinds
• Type **A**: for Rock or stiff deposits of sand
• Type **B**: for deep deposits of medium dense sand, gravel or medium stiff clays.
• Type **C**: Loose cohesionless soil deposits or deposits with soft to medium stiff cohesive soil.

Please refer section 3.2 of Eurocode8 for detailed guidelines regarding the choice of soil type.

**Note:** This is the response spectrum type that is explained in [TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994](on page 2710).

---

**EURO (EC8) - 2004**

**Load Type**
Select either a Elastic or Design response spectrum for the loading type.

Two types of response spectra curve can be generated based on either RS1 (for response spectra type 1 curve) or RS2 (for response spectra type 2 curve).

**Design Ground Acceleration**
Specify a design ground acceleration expressed in terms of acceleration due to gravity (g).

For most of the application of Eurocode 8, the hazard is described in terms of a single parameter (i.e., the value of effective peak ground acceleration in rock or firm soil). This acceleration is termed as the design ground acceleration.

**Behaviour Factor**
Specify the value used to reduce the elastic response spectra to the design response spectra.

The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

**Subsoil Class**
Used to define the subsoil conditions based on which the response spectra will be generated.

Based on the subsoil conditions the soil types may be of three kinds

• Type **A**: rock or other rock-like geographical formation.
• Type **B**: very dense sand, gravel or very stiff clay.
• Type **C**: Deep deposits of dense or medium dense sand, gravel or stiff clay.
• Type **D**: Deposits of loose-to-medium cohesionless soil or of predominantly soft to firm cohesive soil.
• Type **E**: Surface alluvium layer

Please refer section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

**Note:** This is the response spectrum type that is explained in [TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004](on page 2715).

---

**IBC 2006**

**Zip**
The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors.

**Latitude / Longitude**
The geographic coordinates of the site used to determine the Ss and S1 factors. This option may be used if no value is entered for Zip.
### S1 / SS
Mapped MCE for 0.2s spectral response acceleration and spectral acceleration for a 1-second period, respectively. These values may be entered if not geographic coordinate or postal code is provided.

### Long Period (TL)
Long-Period transition period in seconds.

### Fa / Fv
Optional Short-Period site coefficient at 0.2s and Long-Period site coefficient at 1.0s, respectively. Values must be provided if the selected **Site Class (SCL)** is F.

### Site Class (SCL)
Select A through F for the Site Class as defined in the IBC code.

**Note:** This is the response spectrum type that is explained in [TR.32.10.1.8 Response Spectrum Specification per IBC 2006](on page 2743).

### IBC 2012

<table>
<thead>
<tr>
<th>Zip</th>
<th>The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Latitude</strong> / <strong>Longitude</strong></td>
<td>The geographic coordinates of the site used to determine the Ss and S1 factors. This option may be used if no value is entered for <strong>Zip</strong>.</td>
</tr>
<tr>
<td><strong>S1 / SS</strong></td>
<td>Mapped MCE for 0.2s spectral response acceleration and spectral acceleration for a 1-second period, respectively. These values may be entered if not geographic coordinate or postal code is provided.</td>
</tr>
<tr>
<td><strong>Long Period (TL)</strong></td>
<td>Long-Period transition period in seconds.</td>
</tr>
<tr>
<td><strong>Fa / Fv</strong></td>
<td>Optional Short-Period site coefficient at 0.2s and Long-Period site coefficient at 1.0s, respectively. Values must be provided if the selected <strong>Site Class (SCL)</strong> is F.</td>
</tr>
<tr>
<td><strong>Site Class (SCL)</strong></td>
<td>Select A through F for the Site Class as defined in the IBC code.</td>
</tr>
</tbody>
</table>

**Note:** This is the response spectrum type that is explained in [TR.32.10.1.8 Response Spectrum Specification per IBC 2006](on page 2743).

### SNiP II-7-81

**Note:** Only the SRSS and ABS combination methods are valid for the SNiP code.

<table>
<thead>
<tr>
<th><strong>Zoning Factor</strong></th>
<th>Specify the zoning factor per SNiP II-7-81.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Subsoil Class</strong></td>
<td>Defines the subsoil conditions on which the response spectrum will be generated.</td>
</tr>
<tr>
<td>1. Non-weathered rock and rock-like geological formation or permafrost subsoil.</td>
<td></td>
</tr>
<tr>
<td>2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.</td>
<td></td>
</tr>
<tr>
<td>3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.</td>
<td></td>
</tr>
<tr>
<td><strong>Direction</strong></td>
<td>The SNiP code allows an alternate method of specifying directional factors. You may input individual parameters such as KWX, KX1 the product of which is used as the factor along that direction.</td>
</tr>
</tbody>
</table>

**Note:** This is the response spectrum type that is explained in [TR.32.10.1.11 Response Spectrum Specification per SNiP II-7-81](on page 2759).
**NRC 2005**

The NRC 2005 response spectrum has no code-specific parameters.

**Note:** This is the response spectrum type that is explained in [TR.32.10.1.2 Response Spectrum Specification per NRC 2005](on page 2694).

**Related Links**
- [M. To add a generic response spectrum](on page 843)
- [V. NRC 2005 Response Spectrum](on page 3553)
- [V. NRC 2010 Response Spectrum](on page 3563)

**Generated Spectrum** dialog

Displays the generated IBC response spectra data in tabular and graphical format.

**Note:** No data may be modified in this dialog.

Opens when **Generate Spectrum** is clicked on the **Spectrum Parameters** dialog (on page 3027).

- **Spectra Table**: Displays period values with the associated Acceleration, Velocity, or Displacement values for the generated spectra. The table updates based on the Graph Type selection.
- **Graph Paper Type**: Select to display the response spectra period on a Linear or Logarithmic scale.
- **Draw Points**: Select this option to display point markers at each period, displacement/velocity/acceleration coordinate on the curve.
- **Show Peak Value**: Displays the peak displacement/velocity/acceleration in the generated spectrum curve.
- **Graph Type**: Select to display the period versus displacement, velocity, or acceleration on the graph and in the Spectra Table.
- **Close**: Closes the dialog and returns focus to the **Create New Load Item** dialog.

**Related Links**
- [G.17.3.4 Response Spectrum](on page 2371)
- [M. To add an IBC 2000 response spectrum](on page 844)

**Spectrum Parameters** dialog

Used to generate a response spectrum curve per generates the spectrum based on the inputted data and as per section 11.4.5 of the ASCE-7-05 code (which is referenced by the International Building Code).

Opens when the **Generate Spectrum** button is clicked in the **Create New Load Item** dialog from the **Response Spectra** tab (with **Custom** selected for the code).

- **Select Zip**: Type or select the Zip code (US Postal code) to look up the geographic location.
- **Find Lat/Long**: Click to update the Latitude and Longitude values if the postal code value changes.
- **Latitude / Longitude**: Type the geographic coordinates of the site here, if known. These fields will be populated if the Select Zip or Find Lat/Long options are used.
- **Calculate S1 / SS**: Click this button to recalculated the seismic acceleration coefficients if the Latitude and/or Longitude values were manually entered.
S1  This is the mapped MCE spectral response acceleration at a period of 1 second, determined in accordance with section 11.4.1 of ASCE-7-05. It is calculated by setting the Zip code or by clicking the Calculate S1/SS button. Alternatively, you may specify this value directly.

SS  This is the mapped MCE spectral response acceleration at short period, determined in accordance with section 11.4.1 of ASCE-7-05. It is calculated by setting the Zip code or by clicking the Calculate S1/SS button. Alternatively, you may specify this value directly.

Site Class  This is the classification (A to F) assigned to a site based on the soil type as defined in chapter 20 of ASCE-7-05. Values of Fa and Fv from the IBC table are displayed for reference. If the classification is set to F, then you must specify the parameters Fa and Fv.

Fa  Short period site coefficient determined in accordance with table 11.4 -1 of ASCE-7-05. This is determined by the Site Class, but must be specified if the Site Class is set to F.

Fv  Long period (1 sec) site coefficient determined in accordance with table 11.4 -2 of ASCE-7-05. This is determined by the Site Class, but must specified if the Site Class is set to F.

Start / End  Specify the start and end times (T) to define the response time range.

Interval  Specify a time step interval for generating the curve.

Generate Spectrum  Closes the dialog and generates response spectrum curve data. The Generated Spectrum dialog (on page 3027) opens to review the IBC response spectrum data.

Cancel  Closes the dialog without generating response spectrum data.

Related Links
- G.17.3.4 Response Spectrum (on page 2371)
- M. To add an IBC 2000 response spectrum (on page 844)

Repeat Load tab

Used to add load combinations in STAAD.Pro which will be used directly by the analysis engine. Refer to GS. Load Types in STAAD.Pro (on page 47) for additional information.

Repeat Load

Used to create a primary load case using combinations of previously defined primary load cases. A Repeat Load is treated as a new primary load. Therefore, a P-Delta analysis will reflect correct secondary effects. Load Combinations, on the other hand, algebraically combine the results such as displacements, member forces, reactions and stresses of previously defined primary loadings evaluated independently.

Available Load Cases  All primary load cases that are defined for the structure are listed here.

Repeat Load Definition  Loads included in this list make up the Repeat Load.

The Factor cell is used to specify a factor with which the selected primary load case is to be multiplied. The resulting values are utilized in the repeated load definition.
Reference Load
Used to include Reference Loads in a Primary Load case here.

A reference load is defined using the Create Reference Load dialog (on page 3006). Load specifications are then added to the Reference Load entry in the Load & Definition dialog for later reference.

Available Load Cases
All reference load cases that are defined for the structure are listed here.

Referenced Load list
Loads included in this list make up the Reference Load.

The Factor cell is used to specify a factor with which the selected reference load case is to be multiplied. The resulting values are utilized in the primary load definition.

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Include the selected Reference Load Case in the Primary Load Definition list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Include all Reference Load Cases in the Primary Load Definition list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all Reference Load Cases from the Primary Load Definition list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected Reference Load Case from the Primary Load Definition list.</td>
</tr>
</tbody>
</table>

Notional Load
Used to define a Notional Load from Primary Load Cases and / or Reference Load Cases. These are used for direct analysis.

Primary Load Cases
All primary load cases that are defined for the structure are listed here.
Reference Load Cases

All reference load cases that are defined for the structure are listed here.

Notional Load Definition

Loads included in this list make up the Notional Load.

The Factor cell is used to specify the factor with which the selected primary load case or reference load case is to be multiplied. The resulting values are utilized in the notional load definition.

The Direction cell is used to select a global direction in which the (lateral) notional loads act for this case.

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Include the selected Primary Load Case or Reference Load Case, respectively, in the Notional Load Definition list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Include all Primary Load Cases or Reference Load Cases, respectively, in the Repeat Load Definition list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all Primary Load Cases or Reference Load Case, respectively, from the Repeat Load Definition list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected Primary Load Case from the Repeat Load Definition list.</td>
</tr>
</tbody>
</table>

Related Links

- M. To add a repeat load case (on page 872)
- G.17.2.1.4 AISC 360 Direct Analysis (on page 2352)
- M. To add a notional load case (on page 852)
- M. To create a reference load (on page 873)

Frequency tab

Used to specify commands to calculate the frequency using the Rayleigh method or by solving an eigenvalue problem.

Rayleigh Frequency

This option has no additional parameters

Modal Calculation

Consider Missing Mas Mode

Check this option to include the missing mass procedure in the analysis for Steady State/Harmonic analysis.

Related Links

- M. To calculate the structure frequency (on page 834)
Load Generation dialog

Used to create a primary load case using the data of a pre-defined vehicle.

Opens when the Load Generator tool is selected in the Load Generation group on the Loading ribbon tab.

Vehicles definitions are created using the Create New Moving Load Definitions dialog.

![Create New Definitions / Load Cases / Load Items:](image)

**Load Generation**

- No. of Loads to be generated: 1
- Predefined Load to be Added: NONE

**Note:** Once a load definition has been created, the (Moving) Load Generation Type dialog is used to specify the direction and path for the moving loads.

Refer to **TR.32.12.1 Generation of Moving Loads** (on page 2771) for additional details.

**No. of loads to be generated**

The moving load is discretized into a number of distinct positions of the vehicle along the direction of movement. Each position represents a distinct load case whose loads are derived from the corresponding position of the vehicle on the structure. The number of such positions hence has to be communicated in the form of the number of load cases to be generated.

For example, if a bridge is 200 ft long, the first axle to last axle distance of your design vehicle is 15 ft, and you want enough cases to be generated to account for the first axle entering the bridge up to and including the last axle leaving the bridge, that would account for a total distance of 230 ft. If you assume 2 ft increments, then the number of cases would be 230/2 + 1 = 116.

**Predefined load to be added**

With each generated load case, the program Used to include within that load case, other loads from a previous defined primary load case. Such a primary load case might be for example, the dead load case, which the user may defined as load case 1. So, if the user wishes to have each generated load case to consist of a) the selfweight and other dead loads of the structure b) loads resulting from the position occupied on the structure by the vehicle, then, he/she
could specify load case 1 as the predefined load case to be added to each of the generated moving load cases.

**Related Links**

- *G.16.1 Moving Load Generator* (on page 2343)
- *M. To generate moving load cases* (on page 854)

**Define Load Type dialog**

Used to define load types for multiple primary load cases in a single dialog. The load type definition is required if you want the program to automatically generate load combinations from the defined primary loads. Load type and reducible options may be set or editing for primary load cases individually when they are created, as well.

Opens when the **Primary Load Type** tool is selected in the **Load Generation** group on the **Loading** ribbon tab.

### Define Load Type

<table>
<thead>
<tr>
<th>Load</th>
<th>Title</th>
<th>Type</th>
<th>Reducible-IBC 2003</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>DEAD AND LIVE LOAD</td>
<td>None</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>WIND FROM LEFT</td>
<td>None</td>
<td></td>
</tr>
</tbody>
</table>

**Load**

The primary load case number.

**Title**

The name of the primary load case.

**Type**

Select the appropriate load type from the drop-down list in this cell. If the chosen load type is Live or Roof Live, the **Reducible** option is active.

**Note:** If the load type was defined at the time of defining the primary load, the type of the defined load (dead, live, wind etc.) is shown here.
Reducible-IBC 2003
Check this option if you want the live load to be reduced as per the provisions in IBC 2003.

Related Links
•  M. To define primary load type (on page 867)

Auto Load Combination dialog

Used to automatically create load combinations based on code-specified combination rules and primary load type selections.

In order to use this feature, one or more primary load cases must be created with a load type defined. A load type can be assigned to a primary load case either at the time when the load is being created or later.

Opens when the Automatic Combinations > Automatic Load Combinations tool is selected in the Load Generation group on the Loading ribbon tab.

**Note:** If no load cases have yet been defined for the model, a warning message is displayed over the dialog and the parameters are inactive. Primary load cases are defined using the Create Primary Load Case dialog.

**Select Load Combination Code**
The load types available include the following:

- ACI
- AISC
- UBC
- IBC
- British
- NBCC 1995
- Indian Code
- SNiP
- NBC 2005
- IBC 2012

**Note:** Any Load Rule sets you have added or edited in the Edit Load Rules for Auto Load Combination Generator dialog are also available here.

**Select Load Combination Category**
Select the load combination category as specified in the selected load combination code. Refer to the selected code for details.

**Generate Loads**
Click to create all the load combinations. The generated load combinations, with their associated load factors, are shown in the Selected Load Combinations list on the right-hand side.

**Discarded Load Combinations**
Any load combinations which are generated can be removed from the final load combination. These will be listed here.
### List Operators

<table>
<thead>
<tr>
<th>Operator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected primary load case to the Selected Load Combinations list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all primary load cases to the Selected Load Combinations list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all entries in the Selected Load Combinations list, load combination is placed in the Discarded Load Combinations list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected entry from the Selected Load Combinations list.</td>
</tr>
</tbody>
</table>

### Selected Load Combinations List

All load combinations which will be added to the input file are listed here.

### Create Repeat Load Cases

Select this option to create primary load cases for all the load case combinations in the Selected Load Combinations list using the Repeat Load Command, rather than combined results using Load Combinations.

See [GS. Load Types in STAAD.Pro](on page 47) for additional information on using Repeat loads versus Load Combinations.

### Include Notional Load?

Select this option to include Notional Load cases in the generated combinations when the Create Repeat Load Cases option is selected, if they are present in the input file.

### Automatically Generated Load Combinations in the STAAD input file

In the STAAD.Pro Input File, the generated load combinations look like the following:

```
UNIT FEET KIP
LOAD 1 DEAD + LIVE
MEMBER LOAD
2 UNI GY -2.5
LOAD 2 WIND FROM LEFT
JOINT LOAD
2 FX 10
LOAD COMB 3 Generated AISC 1
1 1.6
LOAD COMB 4 Generated AISC 2
1 1.4
LOAD COMB 5 Generated AISC 3
1 1.7 2 0.8
LOAD COMB 6 Manual
1 1.7 2 -1.2
PERFORM ANALYSIS
PRINT MEMBER FORCES
```

The syntax `Generated xxx n` where `xxx` is the design code name (AISC, BOCA, etc.) and `n` is the index in the series of load combinations generated for a specific code, are interpreted by STAAD.Pro. Deleting any of the aforementioned keywords will break the program's ability to read back in the generated load combinations.

The following are some important caveats to keep in mind when using the automatic load combination generator:
• If the automatically generated load combinations are edited (or deleted) from the STAAD input file, they will be reflected in the graphical interface as well.
• The load combination numbers (i.e. LOAD COMB n GENERATED AISC 1) can also be edited from the input file.
• Deleting the keyword GENERATED code index will remove that particular load combination from the automatically generated list. It will not remove the load combination.
• The factors assigned to the primary load cases in a generated load combination cannot be changed in the graphical interface using the **Combine** button from the **Loads** dialog box.

**Related Links**
• [M. To automatically generate load combinations](on page 869)

**Edit Load Rules for Auto Load Combination Generator** dialog

STAAD.Pro includes a facility for the automatic generation of load combination cases. Using the rules described in ACI, AISC, UBC and IBC codes for combining standard load types, you can instruct the program to automatically generate load combination cases from primary load cases which have been assigned load types (i.e., DEAD, LIVE, FLOOD, SNOW, RAIN WATER/ICE, etc.).

You can also use the Edit Loading Rules facility to alter the definitions and factors which come included with the program for these codes or to add new codes containing their own definitions and factors.

Opens when the **Automatic Combinations** > **Edit Auto Combination Rules** tool is selected in the **Load Generation** group on the **Loading** ribbon tab.

### Select Code
Select a predefined or previously created design code from the list to use the load combination rules associated with that code.

### Select Category
Many codes have multiple sets of load combination rules for various cases and structures. Select the category for the selected code from this list. “Category” stands for the various sub-tables we wish to associate with the selected code (e.g., one table for loads on office buildings, another for transmission towers in coastal regions, another for power-plant structures, etc.).

**Note:** Refer to the selected code for additional details regarding the definitions for each category and for which structures and conditions they apply.

### New Code
Opens the Add New Code dialog, which is used to specify a name for a new set of load combination rules, which are typically specified in design codes.

### New Category
Opens the Add New Category dialog, which is used to specify a name for a new load combination table as well as the number of rules (rows) included in this table. New categories can be added to the predefined codes as well as to code rule sets you create. At least one category must be present in each code.

### Add Row
Adds an empty row at the end of the current table (the selected code and category), which can then be populated with a new load combination rule.

### Update Table
Saves changes made to the current table.
Close
Displays the rules associated with the selected code and category.

Rules table
Rows in the table typically represent a single load combination rule (the first two rows are special instructions for how each load type should be handled).

- Include Notional Load? - Select this option for each load type to include Notional Load cases in the generated combinations, if they are present in the input file. These loads are used for direct analysis (on page 3069).
- Combination Rule - As some load types can result in different actions, a single rule may result in multiple load combinations (i.e., wind or seismic acting in positive or negative directions). The combination rule directs the program how to handle such cases:
  - Combine all cases together - For each rule, a single combination will be created which will include all the load cases of that load category multiplied by the factor in the table.
  - Separate combination for each case - For each rule, multiple combinations will be created, each will include one of the load cases of that load category multiplied by the factor in the table.
  - All possible combinations - For each rule, multiple combinations will be created which will include each of the load cases of that load category on their own and with each and every other load case of that category multiplied by the factor in the table.

Examples of each combination rule are below.

- Rule No. - The multiplying factor for each load type for the rule is displayed and may be edited (blue text in a white cell). Load Types with no factor associated with the rule are displayed in yellow cells.

Columns in the table represent load types.

Example 1: Combine All Load Cases
Consider both load categories are set with the option **Combine all load cases together**. This will result in a single load combination:

\[
[(LC1 + LC2 + LC3 + LC4) \times 1.4] + [(LC5 + LC6) \times 1.6]
\]

i.e.,

<table>
<thead>
<tr>
<th>LOAD COMBINATION</th>
<th>Generated Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>1 1.4 2 1.4 3 1.4 4 1.4 5 1.6 6 1.6</td>
</tr>
</tbody>
</table>

Example 2: Separate Combinations for Each Case
Consider the DEAD category now is set to **Combine all load cases together** and the LIVE category is set to **Separate combinations for each case**. For this single rule, this will result in two load combinations due to the two LIVE load cases which are to be considered separately, thus:

\[
[(LC1 + LC2 + LC3 + LC4) \times 1.4] + [(LC5) \times 1.6]
\]

\[
[(LC1 + LC2 + LC3 + LC4) \times 1.4] + [(LC6) \times 1.6]
\]
Example 3: All Possible Combinations

Consider the DEAD category now is set to All possible combinations and the LIVE category is set to Combine all cases together. For this single rule, this will result in 15 load combinations due to the fourteen ways the four dead loads can be combined together and the two LIVE load cases which are to be considered together, thus:

\[
\begin{align*}
\text{(LC1) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC1 + LC2) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC1 + LC2 + LC3) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC1 + LC2 + LC3 + LC4) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC1 + LC3) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC1 + LC3 + LC4) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC1 + LC4) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC2) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC2 + LC3) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC2 + LC3 + LC4) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC2 + LC4) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC3) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC3 + LC4) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\text{(LC4) x 1.4} & + \text{[(LC5 + LC6) x 1.6]} \\
\end{align*}
\]
Related Links

- M. To define automatic load combination rules (on page 867)
- M. To define automatic load combination rules (on page 867)

Create Wind Type Definition dialog

Used to add a new wind load definition.

Opens when the Wind tool is selected in the Define Load Systems group on the Loading ribbon tab.

Type No. Denotes a number by which the wind load type will be identified. Multiple wind types can be created in the same model.

Comments An optional description used to help identify the load.

Related Links

- G.16.3 Wind Load Generator (on page 2345)
- M. To add a wind load definition (on page 835)

Add New Wind Definitions (data) dialog

Used to add wind pressure data or code-specific wind parameters.

Intensity tab

Select Type Choose the method of specifying wind pressure values from the drop-down:

- Custom – ASCE 7 or other user-specified wind pressures

Intensity vs. Height table

Enter pairs of data for wind pressure at a given height (in the current units).

Int cells – wind intensities (pressures) in force/area. Up to 100 different intensities can be defined in the input file per type.

Height cells – corresponding heights in global vertical direction, measured in terms of actual Y (or Z for Z UP) coordinates up to which the corresponding intensities occur.

Calculate as per ASCE 7

Opens the ASCE-7 Wind Load dialog, which is used to generate a wind load per the ASCE 7 specification. Input data on the type of structure, surrounding terrain, and wind.

- SNiP 2.01.07-2011, SNiP 2.01.07-85, or SP 20.13330.2016 – Russian wind load parameters
**Pressure**
the characteristic value of wind pressure, always positive

**Terrain**
terrain roughness category:
- A. Coastal Zone.
- B. Urban Zone.
- C. Large City.

**Classification**
the type of structure:
- 1. Prismatic building structures,
- 2. General-type concrete structures,

**Region**
Wind region as per clause 11.5 of SNiP 2.01.07-85* 2016. This is used to determine the wind pressure in determine the dynamic wind component.

**Delta**
logarithmic decrement of oscillations, Delta (see table 11.5 - Section 11.1.8 for the definition). Typically values are:
- 0.15 for steel towers, masts, lined chimneys, column means including the ones on the reinforced concrete pedestals
- 0.3 for reinforced concrete and stone structures as well as for buildings with steel framework if there are walling structures

**Exposures tab**

**Factor**
exposure factors. A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction. Limit: 99 factors.

**Note:** Note any joint does not have an exposure assigned, the exposure will be taken as 1.0.

**Related Links**
- 6.16.3 Wind Load Generator (on page 2345)
- M. To add a wind load definition (on page 835)
- M. To add an ASCE 7 wind load definition (on page 836)
- M. To add a SNiP wind load definition (on page 837)

**ASCE 7 Wind Load dialog box**

Used to generate a wind load per the ASCE 7 specification. Input data on the type of structure, surrounding terrain, and wind.

Opens when **Calculate as per ASCE-7** is clicked on the **Add New: Wind Definitions** dialog box (on page 3018) when **Custom** is selected as the type..

Using this feature, you can define different intensity vs. height data for the windward side, leeward side and side walls for a structure. Each can be defined as a different load type and then applied in the relevant direction with appropriate direction factor. The defined wind loading can then be applied to the structure following the usual method.
**Note:** For details on generation of wind load and command syntax, refer to TR.32.12.3 Generation of Wind Loads (on page 2779).

**Common tab**

**ASCE-7-**

**Building classification category**
Select the building classification category, based on the nature of occupancy, as obtained from Table 1-1 in ANSI/ASCE 7-95 or SEI/ASCE 7-02.

Category can be I, II, III or IV.

**Basic Wind Speed**
Specify a basic wind speed as described in Figure 6-1 in the the ANSI/ASCE 7-95 code or in section 6.5.4 of the SEI/ASCE 7-02 code, along with units.

Wind speed is assumed to be the plotted or tabulated 3-second gust wind speed.

**Exposure Category**
Select an exposure category as per section 6.5.3 in the ANSI/ASCE 7-95 code or section 6.5.6.3 of the SEI/ASCE 7-02 code.

**Structure Type**
Select the type or structure that best fits the model from the following:

- Building structures
- Chimneys, Tanks, and similar structures
- Solid Signs — Used to generate wind loads on solid signs or walls and calculates the Cf factor per the selected code version.
- Open Signs
- Latticed Framework
- Trussed Tower

**Note:** The data input tab changes to the selected structure type once the **Apply** button is clicked.

**Consider Wind speed-up over hill or escarpment**
Select **Yes** to consider wind speed-up over a hill or an escarpment and **No** to ignore it.

If there are isolated hills and escarpments that constitute abrupt changes in the general topography, the increase in speed can be considered as per clause 6.5.5 in the ANSI/ASCE 7-95 code or as per section 6.5.7 in the SEI/ASCE 7-02 code.

**Type of Hill or Escarpment**
Select the type of hill, ridge or escarpment on which the structure is located, based on Figure 6-2 of ANSI/ASCE-7-95 code or on Figure 6-4 of the SEI/ASCE 7-02 code. The options available are 2-D Escarpment, 2-D Ridge and 3-D Axisymmetric Hill.

**Height of Hill or Escarpment (H)**
Specify the height of the hill or escarpment relative to the upwind terrain (H in Figure 6-2 of the ANSI/ASCE-7-95 code or in Figure 6-4 of the SEI /ASCE-7-02 code).

**Distance upwind of crest (L_h)**
Specify the distance upwind of crest to where the difference in general elevation is half the height of the hill or escarpment (L_h in Figure 6-2 of the ANSI/ASCE-7-95 code or in Figure 6-4 of the SEI/ASCE-7-02 code).

**Distance from the crest to the building (x)**
Specify the distance from the crest to the building site. A negative value signifies the distance is in the downwind direction (X in Figure 6-2 of the ANSI/ASCE-7-95 code or in Figure 6-4 of the SEI/ASCE-7-02 code).
<structure type> tab

When the structure type is changed on the Common tab, the list of parameters on the Main Building tab changes. The full list of parameters explained here is for the structure type of **Building Structures**. Those parameters that are different are explained below for each type of structure.

Structure Type: Building Structures

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Building Height</td>
<td>Specify the height above ground to the highest point on the roof surface, along with units.</td>
</tr>
<tr>
<td>Building Length along the direction of Wind (L)</td>
<td>Length of building measured along the direction of wind.</td>
</tr>
<tr>
<td>Building Length Normal to the direction of Wind (B)</td>
<td>Length of building measured normal to the direction of wind.</td>
</tr>
<tr>
<td>Building Natural Frequency</td>
<td>Specify the natural frequency of the building to calculate Gust Effect Factor.</td>
</tr>
<tr>
<td>Building Damping Ratio</td>
<td>Specify the damping ratio to calculate Gust Effect Factor.</td>
</tr>
<tr>
<td>Enclosure Classification</td>
<td>For ASCE 7-02, select the building classification as open, partially enclosed, or enclosed per the provisions of section 6.2 of SEI/ASCE 7-02.</td>
</tr>
</tbody>
</table>

**Note:** Not applicable for ASCE 7-95.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kz</td>
<td>Velocity Pressure exposure coefficient calculated as per Table 6-3 of the ANSI/ASCE-7-95 code or as per Table 6-3 of the SEI/ASCE-7-02 code.</td>
</tr>
<tr>
<td>Use Kzt</td>
<td>Select this option to specify a topographic factor value for wind speedup.</td>
</tr>
<tr>
<td></td>
<td>If not selected, the topographic factor is calculated and displayed. When wind speedup is considered, Kzt is calculated as per Eq 6-2 of the ANSI/ASCE-7-95 code or as per Eq 6-4 of the SEI/ASCE-7-02 code.</td>
</tr>
<tr>
<td>Use I</td>
<td>Select this option to specify an importance factor value.</td>
</tr>
<tr>
<td></td>
<td>If not selected, the importance factor is calculated and displayed. The importance factor used is as per Table 6-2 of the ANSI/ASCE-7-95 code or as per section 6.2 of the SEI/ASCE-7-02 code.</td>
</tr>
<tr>
<td>Use Kd</td>
<td>Select this option to specify a wind directionality factor.</td>
</tr>
<tr>
<td></td>
<td>If not selected, the wind directionality factor is calculated and displayed. The wind directionality factor is calculated as per Table 6-4 of the SEI/ASCE 7-02 code.</td>
</tr>
</tbody>
</table>

**Note:** Not applicable for ASCE 7-95.

Structure type: Chimney, Tank and similar Structures

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height(H)</td>
<td>Height of the structure as defined by the term “h” in Figure 6-19 of the SEI/ASCE 7-02 code.</td>
</tr>
</tbody>
</table>

**Note:** Not applicable for ASCE 7-95.
**Least Horizontal Dimension (W)**
Smaller of the plan dimensions. In case the cross section of the structure in plan is circular, the diameter needs to be specified.

**Horizontal Cross-Section Type**
This is the cross section of the structure in plan as defined in Figure 6-19 of the SEI/ASCE 7-02 code. The available options include square with wind being normal to face or acting along the diagonal, hexagonal, octagonal and round.

**Depth of protruding elements such as ribs and spoilers (D')**
For round type cross sections, depth of protruding elements need to be defined which is a measure of the surface roughness as indicated in Figure 6-19 of the SEI/ASCE 7-02 code.

**Structure type: Solid Signs**

**Height (H)**
Height of the structure which is used for calculating the height to width ratio as defined by the term "ν" in Figure 6-20 of the SEI/ASCE 7-02 code.

**Horizontal Dimension of Sign (M)**
Horizontal dimension of the solid sign

**Vertical Dimension of Sign (N)**
Vertical dimension of the solid sign.

*Note:* If the sign is at the ground level, the height (H) and vertical dimension (N) should both be specified the same value.

**Structure type: Open Signs / Lattice Frame Work**

**Ratio of Solid Area to Gross Area**
Ratio of solid area to gross area as indicated by the term $\epsilon$ in Figure 6-21 of the SEI/ASCE 7-02 code.

**Orientation of the members exposed to wind**
The type of member surfaces which are exposed to wind. Select flat-sided members or rounded members in Figure 6-21 of the SEI/ASCE 7-02 code.

**Diameter of typical round member**
Diameter for round members as defined by the term “D” in Figure 6-21 of the SEI/ASCE 7-02 code.

**Structure type: Trussed Tower**

**Horizontal Cross Section**
The type of cross section of the tower in plan as defined in Figure 6-22 of the SEI/ASCE 7-02 code. The available options include square and triangle.

**Building Design Pressure tab**

**Building Wall to generate Wind Load on:**
Select the side of the structure, with respect to wind direction, for which you wish to generate a wind load. The pressure will be calculated as per Table 6-1 of the ANSI/ASCE-7-95 code or as per Equation 6-23 of the SEI/ASCE-7-02 code, based on the code selection made on the tab. The relevant equation will also be displayed below.

- Windward - Used to generate the design wind pressure for the windward side.
- Leeward - Used to generate the design wind pressure for the side wall.
- Side Wall - Used to generate the design wind pressure for the side wall.

*Note:* The pressure profile for each of the building walls has to be individually determined under a unique load type number. Thus, generating the profile for the three sides of the building constitutes three separate steps and thus, three separate types. Each type can then be
applied with one load case or separate load cases and then applied in the relevant direction with the appropriate direction factor. Examples illustrating wind load generation can be found in the examples manual.

**Use G**
Select this option to specify a gust effect factor value.

If not selected, the Gust effect factor is calculated and displayed. The gust effect factor is calculated as per Table 6-1 of the ANSI/ASCE-7-95 code or as per section 6.5.8 of the SEI/ASCE-7-02 code, based on the code selection made on the Common tab.

**Use C_p**
Select this option to specify a wall pressure coefficient value.

If not selected, the wall pressure coefficient is calculated and displayed. The wall pressure coefficient is calculated as per as per Fig 6-3 of the ANSI/ASCE-7-95 code or as per Figures 6-11 through 6-17 of the SEI/ASCE-7-02 code, based on the code selection made on the Common tab.

**Use G_Cpi**
Select this option to specify an internal pressure coefficient value.

If not selected, the internal pressure coefficient is calculated and displayed. The internal pressure coefficient is calculated as per per Table 6-4 of the ANSI/ASCE-7-95 code or as per Figure 6-5 of the SEI/ASCE-7-02 code, based on the code selection made on the Common tab.

**Use C_f**
Select this option to specify a force coefficient value used for structures other than buildings. If not selected, the force coefficient is calculated and displayed as follows:

- For Chimney, Tank, and similar structures: Calculated as per Figure 6-19 of the SEI/ASCE 7-02 code.
- For Solid signs: Calculated as per Figure 6-20 of the SEI/ASCE 7-02 code.

The force coefficient is used for the calculation of the design pressure.

**Height vs. Intensity table**
Displays the generated wind profile based on the current input in the dialog.

**OK**
Accepts the current Height vs. Intensity table values displayed on the Building Design Pressure tab and applies them to the Add New: Wind Definitions dialog.

**Apply**
Applies changes made in one of the dialog tabs to other tabs for use (e.g., the code selection was changed).

**Cancel**
Closes the dialog without adding an ASCE wind definition.

**Help**
Opens the STAAD.Pro Help window.

**Related Links**
- [M. To add an ASCE 7 wind load definition](on page 836)
- [M. To add an ASCE 7 wind load definition](on page 836)

**Add New Snow Definition dialog**

Used to define parameters used for generating snow loading on a structure per the ASCE 7-02 code.

Opens when the Snow tool is selected in the Define Load Systems group on the Loading ribbon tab.
The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

As a result, the members for which the snow load is to be generated must be clustered together into Floor Groups. For details on creation of groups, refer to TR.16.1 Listing of Entities by Specifying Groups (on page 2440).

**Type No:** Specify an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model.

**Ground Snow Load** The pressure or, weight per unit area, to be used for the calculation of the design snow load, in current units. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of the SEI/ASCE 7-02 code.

**Exposure Factor** Specify the exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.

**Thermal Factor** Specify the thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.

**Importance Factor** Specify the snow importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

**Related Links**
- [M. To add an ASCE 7-02 snow load](on page 851)

## Add New Seismic Definitions dialog

Used to define the parameters for performing a dynamic analysis using the static equivalent approach as outlined in the various seismic codes supported by STAAD.Pro.

Opens when the Seismic > <code name> tool is selected in the Define Load Systems group on the Loading ribbon tab.

Weights are added to a seismic definition through the other tabs of the Add New: Seismic Definition dialog. Seismic load items using this definition are applied using the Add New: Load Items dialog.

### Seismic Parameters tab

**Type** Select the code you wish to use for defining seismic parameters.
- [UBC 1985 or 1994](on page 2617)
- [UBC 1997](on page 2620)
- [IBC 2000 or 2003](on page 2588)
- [IBC 2006](on page 2592)

**Note:** Added in STAAD.Pro 2007 (Build 03). Refer to AD.2007-03.1.6 IBC 2006 in the What’s New section for additional information.

- [IBC 2012](on page 2596)
Included in STAAD.Pro 2007 (Build 11). Refer to the What’s New section for additional information.

- **IS 1893 - 2002/2005** (on page 2576)
- IS 1893 - 1984
- **Canadian NRC 1995** (on page 2548)
- **Canadian NRC 2005** (on page 2552)
- **Mexican CFE 1987 or 1993** (on page 2607)
- **Mexican NTC** (on page 2610)
- **Japanese (AIJ)** (on page 2605)
- **Colombian** (on page 2571)
- **Turkish** (on page 2613)
- **Algerian** (on page 2546)
- **Chinese GB 50011-2001** (on page 2563)
- **Colombian 2010** (on page 2573)

**Include Accidental Load**

Select this option to calculate the accidental torsion component described in the appropriate code.

**Include IS 1893 Part 4**

(IS 1893 - 2002/2005 only) Set this option to specify loads per IS 1893 (Part 4) 2005 for industrial or stack-like structures. When set, the a value for ST is required (1 or 5). When set and ST of 5 is used, CV and DV are also required.

**Generate Parameters table**

(IS 1893 - 2002/2005 only) Opens the **IS:1893 Seismic Parameters** dialog box, which is used to populate the parameter fields with appropriate values for the IS 1893 - 2002 code.

A table of the code-relevant seismic parameters is listed. Refer to **TR.31.2 Definitions for Static Force Procedures for Seismic Analysis** (on page 2544) for technical details on each code parameter.

**Self Weight tab**

Used to add the selfweight of the structure.

**SelfWeight Factor**

The factor to be used to multiply the selfweight.

**Joint Weights tab**

Used to add the concentrated weights acting at one or more joints.

**Joint Weight**

Specify the magnitude of the joint weight.

**Member Weights tab**

Used to add distributed and concentrated weights acting on member spans.

**Loading Type**

Select either a **Concentrated** or **Uniform** load type.
**Weight**
Specify the intensity of the distributed weight or magnitude of the concentrated weight.

**Starting / Ending Distance**
Specify the location along the member where the weight is applied, in current units.

---

**Element Weights tab**
Used to add pressure loads on slabs if the structural model consists of plate elements representing entities like floor slabs.

**Pressure**
Specify the magnitude of the uniform pressure, in the current units. Since it is a weight, it is a quantity without a sign.

---

**Reference Load tab**
Used to add reference loads to the seismic definition.

**Available Load Cases**
A list of all previously defined reference loads included in the STAAD.Pro input file. Reference loads are created using the Create Reference Load dialog (on page 3006).

**Referenced Load**
A list of included reference load cases for to be used in the seismic definition are included here. You may specify a different **Factor** value by which the reference load case is multiplied when used for the seismic load.

**Along**
Select a global direction along which the reference load case acts as a weight.

---

**Floor Weights tab**
Used to add floor loads when no slab is present or defined in the structural model.

For additional information on the Floor Load command, refer to TR.32.4.3 Floor Load Specification (on page 2672). The parameters for Floor Weight are analogous to those for the Floor Load command.

**Tip:** If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better. See illustrative example at the end of this section

---

**Range**
Select this option to specify a Floor load or One-way load by specifying a range.

**Group**
Select this option to specify a Floor load or One-way load to act on a previously defined Floor group.

Groups are created using the Create Group dialog (on page 3099).

**Pressure**
Specify the magnitude of the uniform pressure, in the current units. The pressure value is provided as a quantity without sign because it is contributing to the overall weight - a numerically positive term.

**Define Y/X/Z Range**
Specify **Minimum** and **Maximum** values to define a range in the global directions when the Range option is selected.

**One Way Distribution**
Set this option to use a one-way load distribution. Refer to TR.32.4.2 One-way Load Specification (on page 2666) for additional information on using a one-way load.
Towards (One-way Distribution only) Select an existing member onto which the loading is directed and defines the span direction for the one way loading. Otherwise, the load will be distributed to the longer member.

Member Group Select a previously defined floor group from the list.

Inclined (Group option only) This option is required to be set when a group of members that form a panel are inclined to the global XY, YZ, or ZX planes.

Related Links
- M. To add a seismic load definition (on page 840)

**IS:1893 Seismic Parameters dialog box**

Used to populate the parameter fields with appropriate values for the IS 1893 - 2002 code.

Opens when Generate is clicked in the Seismic Parameters dialog (on page 3044) with the IS 1893 - 2002 code selected.

Refer to TR.31.2.9 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2576) for additional information on all parameters.

- **Zone Factor Choice** In the first drop-down list, select either City or Zone. The second drop-down list is then populated with a list of major Indian cities or seismic zones. The corresponding zone factor, Z, is displayed for the current selection.

- **Response Reduction** Select the type of lateral load resisting system, as defined by the IS 1893 code, used in the structure for the direction of loading.

- **Importance Factor** Select the importance classification, as defined by the IS 1893 code.

- **Rock/Soil Type** Select the soil classification for the structure site.

- **Structure Type** Select the material used for the structure.

- **Damping Ratio** Type a damping ratio, in percent.

- **Foundation Depth** Set this option to display a field in which to type the depth of foundation below ground level, in the current units of length.

- **Period in X (sec)** Set this option to display a field in which the period of structure (in sec) in X direction.

- **Period in Z (sec)** Set this option to display a field in which the period of structure (in sec) in Z direction.

- **Generate** Closes the dialog box and populates the values in the Seismic Parameters dialog box.

- **Cancel** Closes the dialog box without generating the values for the Seismic Parameters dialog box.

**Add New Direct Analysis Definition dialog**

Used to define parameters used in the direct analysis method described in AISC 360 Appendix 7.

Opens then the Direct Analysis tool is selected in the Define Load Systems group on the Loading ribbon tab.

**Note:** This feature is used in conjunction with the Perform Direct Analysis command.

Refer to TR.31.7 Definition of Direct Analysis Members (on page 2643) for additional details.
Each tab in the dialog contains a different parameter used for direct analysis. The parameters are:

**FLEX**  Identification of members whose flexural stiffness is considered to contribute to the lateral stability of the structure, along with the initial value of $\tau_b$ that should be used. Members listed with FLEX will have their EI factored by 0.80 times $\tau_b$ while performing the global solution. The final member forces and code check will be with 100% of the flexural stiffness.

**AXIAL**  Identification of members whose axial stiffness is considered to contribute to the lateral stability of the structure. These members will have their EA factored by 0.80 while performing the global solution. The final member forces and code check will be with 100% of the axial stiffness.

**FYLD**  Used to indicate the yield stress of the members this is used only in the calculation of $\tau_b$.

Related Links

- [G.17.2.1.4 AISC 360 Direct Analysis](on page 2352)
- [M. To define direct analysis parameters](on page 852)

### Add New Vehicle Definitions dialog

Used to define different types of moving loads. Several load cases can be generated by applying these types of loads.

Opens when the **Vehicle** tool is selected in the **Define Load Systems** group on the **Loading** ribbon tab.

In order to generate a set of static loads, due to the movement of the vehicle or load on the structure, there are two steps involved. The first step is to define the vehicle. The second step involves the **Moving Load Generator** (on page 3031).

#### Define Load tab

Used to define a moving load system in the input file.

**Vehicle Type Ref**  This is the reference number.

**Width**  Specify a wheel spacing, if applicable. The **Width** is the spacing between loads perpendicular to the direction of movement. If omitted, one directional loading is assumed.

**Load, Dist table**  Enter the value of the concentrated loads and the distance between them. The first load is used as the Reference load for vehicle placement in the (**Moving**) Load Generation Type dialog.

#### AASTHO Spec tab

Used to specify a standard AASHTO load in the input file.

**Vehicle Type Ref**  This is the reference number.

**AASHTO Specification**  Choose from the following standard AASHTO loading: HS20, HS15, H20 and H15.

**Factor**  Specify a multiplying factor, if applicable. The default factor is 1.0.

**Variable Spacing**  (For use with HS loads only) Specify a value for a variable wheel spacing between the second and third axles, per Fig. 3.7.7A of AASHTO Standard Specification or Cl. 3.6.1.2.2
of the AASHTO LRFD. Valid range from 14 feet to 30 feet (inclusive), with the default being 14 feet if not specified.

**File Input tab**

Used to specify a moving load defined in an external file.

**Vehicle Type Ref** This is the reference number.

**File Name** Name of the external file containing moving load data. File name should be limited to 16 characters, reside in the same directory as the current input and have no file extension.

For more information on creating an external moving load file, see TR.31.1 Definition of Moving Load System (on page 2541).

**Load Name** Name assigned to the moving load system in the external file.

**Factor** Specify a multiplying factor, if applicable.

**Related Links**
- G.16.1 Moving Load Generator (on page 2343)
- M. To define a vehicle for loading (on page 853)

**Add New : Pushover dialog**

Used to define structural parameters for a pushover analysis.

Opens when the Pushover tool is selected in the Define Load Systems group on the Loading ribbon tab.

**Common Dialog Controls**

- **Add** Adds the parameters set on the current tab to the input file.
- **Close** Closes the dialog.
- **Help** Opens the STAAD.Pro help window.

**Define Input tab**

Used to define general definition and member specific parameters for a pushover analysis.
Figure 331: Add New : Pushover dialog Define Input tab

General Input Parameters
Select this control to add general parameters for the entire structure.

Type of Frame
Select either a fully restrained **Moment Frame (FR)** or a (Concentric) **Braced Frame (CBF)** to define the type of steel frame.

Geometric Non-Linearity Effect
Select either **Ignore Effect**, if nonlinear effects are not to be considered, or **Include Effect**, if nonlinear effects are to be considered.

Displacement Tolerance
(For when Geometric Non-Linearity Effect is considered only) Set this option to specify a displacement tolerance for convergence, in current input units.

No. of Iterations
(For when Geometric Non-Linearity Effect is considered only) Set this option to specify a target number of iterations for geometric nonlinearity convergence.

No. of Iterations for KG Matrix
Set this option to specify the number of iterations to be performed for geometric stiffness, KG, matrix. The value is an integer, with a maximum value of 50.

Max. no. of Analysis Cycles
Set this option to specify a maximum number of cycles to be considered during the strain hardening stage for the load increment stage.

Displacement Incremental Value
Set this option to include intermediate displacement increments, in current units of length, at which the program will record analysis results for post-processing.
Base Shear Incremental Value
Set this option to include intermediate base shear increments, in current units of force, at which the program will record analysis results for post-processing.

Print Output Results
Set this option to print analysis results for the final state for a nonlinear static analysis.

- Select the Print Step Results option to include the analysis results of the final load step in the output.
- Select the Print Results Output option and then a results type to limit the output.

Member Specific Parameters
Select this control to add parameters for members which will be assigned later.

Expected Yield Stress
Expected yield strength of the material based on Section 5.3.2.3 and Tables 5-1 and 5-2 of FEMA 356.

Effective Length Factor for Member (Y/Z)
The effective length factor, k, defined for any member in either the local Y or local Z direction.

Related Links
- M. To define general pushover data (on page 862)
- M. To define member specific pushover data (on page 862)
- M. To define member specific pushover data (on page 862)
- G.17.4.2.1.1 Define Steel Moment and Braced Frames (on page 2386)
- M. To define general pushover data (on page 862)
- M. To define general pushover data (on page 862)
- M. To define general pushover data (on page 862)
- M. To define general pushover data (on page 862)
- M. To define general pushover data (on page 862)
- M. To define general pushover data (on page 862)

Define Loading Pattern tab
Used to define general definition and member specific parameters for a pushover analysis.
**Loading Pattern**
Select either **Auto** or **User Defined** method for specifying loading patterns.

**Method for Lateral Load Calculation**
(Auto loading pattern only) Set this option to specify the method used (on page 2385) for defining the load pattern.

**Total Base Shear To Be Distributed**
(Auto loading pattern only) Set this option to define base shear that will be distributed vertically along the height of the structure at each floor level. If no value is specified, the program distributes 10 percent of gravity loading as lateral load.

**Direction**
(Total base shear distributed only) Select the global direction in which the pushover load acts.

**Total Base Shear**
(Total base shear distributed only) Base shear value in current units of force used for loading.

**Number of Push Load Steps**
The number of load steps. The lateral load at each floor divided by the number of load steps gives the push load increment at that floor level.

**Related Links**
- *M. To add a pushover loading* (on page 864)
- *M. To add a pushover loading* (on page 864)
- *M. To add a pushover loading* (on page 864)
- *G.17.4.2.1.3 Define Lateral (Push) Loading* (on page 2387)
- *G.17.4.1.8 Lateral Load Distribution* (on page 2385)
- *M. To add a pushover loading* (on page 864)
Define Spectrum Details tab

Used to define the response spectrum according to FEMA 356.

**Critical Damping (%) 1st | 2nd | 3rd | 4th Spectrum**
Specify critical damping ratios to be used for the first (required, 0.05 is the default) through the fourth spectra.

**Site Category**
Specify the site category which describes the soil conditions.

**At Short Period**
Specify the mapped spectral acceleration at short period, $S_s$.

**At One-Second Period**
Specify the mapped spectral acceleration at one-second period, $S_1$.

**Related Links**
- *G.17.4.2.1.7 Define Input for Demand Spectrum* (on page 2390)
- *M. To define pushover spectral data* (on page 863)

Define Hinge Property tab

Used to define the hinge type used in the pushover analysis and create custom hinge definitions.
**Ribbon Control Reference**

**Loading tab**

**Figure 334: Add New : Pushover dialog Define Hinge Property tab**

**Hinge Type**
Select either
- **FEMA** — instructs the program to use the built-in hinge definitions.
- **Ignore** — instructs the program to ignore hinge formation in some members.
- **User Defined** — used to define a custom hinge by **Load Deformation Curve Points**, **Yield Moment**, **Yield Rotation**, and **Acceptance Criteria**.

**Type IDs**
(User defined type only) Select an existing hinge type ID number or select **New Type** to define a new hinge type.

**Type Identifier**
(User defined type only) Specify an unique ID number for the hinge type (hinge property number).

**Load Deformation Curve Points**
Points on the load deformation curve at points A, B, C, D and E as specified in FEMA. The Xi and Yi coordinates of the load deformation curve represent the Deformation Ratio and Moment Ratio, respectively. The coordinates of A and B are preset at 0,0 and 1,1, respectively, to normalize the ratio curve.

**Yield Moment (YM)**
Yield moment in current force units.
Yield Rotation (YR)  
Yield rotation in radians.

IO  
Acceptance criteria limit (deformation ratio) for Immediate Occupancy level.

LS  
Acceptance criteria limit (deformation ratio) for Life Safety level.

CP  
Acceptance criteria limit for (deformation ratio) Collapse Prevention level.

Related Links
- M. To manually define and assign hinges (on page 863)
- G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria (on page 2388)
- G.17.4.1.4 Types of Nonlinearity (on page 2381)
- G.17.4.1.6 Frame element hinge properties (on page 2382)
- M. To manually define and assign hinges (on page 863)

Define Solution Control tab

Used to define the limit of the pushover analysis by either limiting base shear or displacement at a control joint.

![Figure 335: Add New : Pushover dialog Define Solution Control tab](image-url)
**Push Up to Defined Base Shear**

Set this option to limit the analysis by force. The pushover analysis will continue until the cumulative base shear is less than or equal to the base shear specified by this command or the structure has additional strength.

**Direction**

Select the global direction of base shear for this solution control.

**Defined Base Shear**

Specify a target base shear value, in current units of force.

**Push Up to Defined Displacement at Control Joint**

Set this option to limit the analysis by displacement. The pushover analysis will continue until the displacement at the specified joint at the specified direction exceeds specified displacement.

**Direction**

Select the global direction of displacement for this solution control.

**Joint Displacement Value**

Specify a target displacement value, in current units of length.

**Joint Number**

Select the control joint from the list of nodes.

**Related Links**

- [G.17.4.2.1.6 Define Pushover Analysis Solution Control](on page 2390)
- [M. To define solution control](on page 865)

**Add New Time History Definitions** dialog

Used to define the Forcing Function of a time varying load.

Opens when the **Time History > Forcing Functions** tool is selected in the **Dynamic Specifications** group on the **Loading** ribbon tab.

**Note:** Refer to [TR.31.4 Definition of Time History Load](on page 2630) for additional information.

**Note:** After defining a Time History Load, use the **Create New Load Items** dialog **Time History** tab to apply the load to the structure.

**Integration Time Step**

Solution time step used in the step-by-step integration of the uncoupled equations.

**Consider Missing Mass Mode**

Check this option to include the missing mass procedure in the time history analysis.

**Type**

This refers to the number of the type of functions.

**Loading Type**

Select the **Acceleration**, **Force**, or **Moment** option to define the type of function being input.

**Save**

Select this option to create an external file (input file name with a `.tmh` extension) containing the history of the displacements of every node of the structure at every time step.

The following function type options are available:
Select the Define Time vs. Acceleration, Harmonic Function or From External File option and provide function data.

**Define Time vs. <Loading Type>**

Used to specify a time history forcing function, where the loading type is that selected above (either Moment, Force, or Acceleration).

Specify the values Time and corresponding Force or Acceleration.

**Note:** The time history forcing function is plotted on the graph at the bottom of the dialog as data pairs are entered.

**From External File**

If the From External File option is selected, the File Name edit box becomes active.

Type the name of the external file containing time history data. File name should be no more than 8 characters, reside in the same directory as the current input and have no file extension. The data must be provided as one time-force or time-acceleration pair per line.

**Note:** For more information on creating an external time history file, see TR.31.4 Definition of Time History Load (on page 2630).

**Harmonic Function**

To specify a time history harmonic function, check the Harmonic Function button.

**Curve Shape** Specify if the harmonic function is a SINE or COSINE curve.

**Frequency or RPM** Choose Frequency and enter a circular frequency in cycles per second or RPM and enter revolutions per minute.

**Amplitude** Max. Amplitude of the forcing function in current units.

**Phase** Phase Angle in degrees.

**Cycles** No. of cycles of loading.

**Step or SubDiv** Choose the Step option to time step of loading or SubDiv to subdivide a ¼ cycle into this many integer time steps.

**Spectrum**

Select this Function Option to provide Spectrum parameters for your time history loading.

**Tmax** The maximum time (in seconds) in the generated time history.

**DeltaT** Difference (delta) value in subsequent time steps in the generated time history.

**Damp** Damping ratio (5% is entered as 0.05) associated with the input spectrum.

**T1 / T2 / T3** Ending time of the acceleration rise, stead acceleration, and acceleration decay, respectively (in seconds).

**Random Seed** Enter a positive integer (1 to 2147483647) to be used as a unique random number generation “seed.” A unique time history will be produced for each seed value. Change this
value when you want to produce a "different (from the time history generated with the prior seed value)" but statistically equivalent time history.

No of digitized Freq.  The input shock spectrum will be re-digitized at this number of equally spaced frequencies by interpolation. The default is taken as the greater of 35 or the number of points in the input spectrum.

No. of Iteration  The number of iterations which will be used to perfect the computed time history.

Related Links
- G.17.3.5 Response Time History (on page 2373)
- M. To define a time history type from tabular data (on page 855)
- M. To define a time history type from a function (on page 856)
- M. To define a time history type by spectrum (on page 857)
- M. To define a time history type by external file (on page 859)

Define (Time History) Parameters dialog

Used to define time step, damping, and arrival times for time history loads.

Opens when the Time History > Parameters tool is selected in the Dynamic Specifications group on the Loading ribbon tab.

Time Step  Specify a solution time step to be used in the step-by-step integration of the uncoupled equations. The default value of Time Step is determined as explained in TR.31.4 Definition of Time History Load (on page 2630).

Damping  The following options are available for specifying damping:

- **Damping** – This is to be used for specifying a single modal damping ratio which will be applied to all modes. The default value is 0.05.
- **CDAMP** - If a damping ratio has already been specified under CONSTANTS based on the type of material in the structure, that value may be used directly in the time history analysis. Check this option for that purpose.
- **MDAMP** - In we wish to utilize individual damping ratios for individual modes, that is achieved through the means of the MDAMP option. The first step to doing this is the specification of those individual damping ratios, as explained under TR.26.4 Modal Damping Information (on page 2510), and is done graphically by using the Modal Damping tool on the Loading ribbon tab. If this first step has been completed, the instruction to utilize MDAMP is done by selecting this option shown above.

Arrival Time  Specify values of possible arrival times of the various dynamic load types. The arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load may have different arrival times for different joints and hence all these values must be specified here. The arrival times and time-force pairs for the load types are used to create the load vector needed for each time step of the analysis.
Add
Adds the specified load details to the currently selected load case in the Load dialog and closes the dialog.

Close
Closes the dialog without adding any load data.

Help
Opens the STAAD.Pro Help window.

Related Links
• G.17.3.5 Response Time History (on page 2373)
• M. To define time history parameters (on page 860)

Modal Damping dialog

Used to define unique damping ratios for the individual modes used in a dynamic analysis. It is used in response spectrum and time history analysis.

Opens when the Modal Damping tool is selected in the Dynamic Specifications group on the Loading ribbon tab.

Note: STAAD.Pro is also capable of calculating each mode's damping based on the frequency of the mode, the mass-proportional damping coefficient (α), and the stiffness-proportional damping coefficient (β). This input must be provided using the STAAD.Pro Editor. Refer to Calculate Damping (on page 2368) for additional details.

Evaluate
This option is to instruct STAAD.Pro to calculate the damping ratio for each mode based on the frequency of the mode and the Minimum and Maximum values entered in the respective fields.
The formula used to evaluate the damping ratio is explained in Evaluate Damping (on page 2370).

**Explicit**  
Provide values of damping ratio for the corresponding mode number.  
Refer to Explicit Damping (on page 2368) for additional details.

**Related Links**
- G.17.3.3.2 Modal Damping (on page 2368)
- M. To explicitly define damping values for modes (on page 875)
- M. To evaluate damping for modes (on page 875)

## Analysis and Design tab

### Table 320: Analysis Data group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Analysis Commands" /></td>
<td>Opens the Analysis/Print Commands dialog for analysis commands, which is used to define the analysis commands to be included in the Input Command File.</td>
</tr>
<tr>
<td><img src="image" alt="Pre Analysis Commands" /></td>
<td>Opens the Analysis/Print Commands dialog for pre-analysis print commands, which is used to define the pre-analysis print commands to be included in the Input Command File.</td>
</tr>
<tr>
<td><img src="image" alt="Post Analysis Commands" /></td>
<td>Opens the Analysis/Print Commands dialog for post-analysis print commands, which is used define the post-analysis print commands to be included in the Input Command File.</td>
</tr>
<tr>
<td><img src="image" alt="Input Width" /></td>
<td>Opens the Input Width dialog, which is used to set the maximum allowable number of characters per line in the input command file for processing. Choose 72 or 79 columns and then click OK. Refer to TR.4 Input/Output Width Specification (on page 2413) for additional information.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| Output Width        | Opens the **Output Width** dialog, which is used to set the maximum allowable number of characters per line in the output file.  
                    | Choose 72 or 118 columns and then click **OK**.  
                    | **Note:** If you choose 118 Columns, your printer must have the capability of supporting that wide of an output.  
                    | Refer to [TR.4 Input/Output Width Specification](#) (on page 2413) for additional information.                                                                 |
| Floor Diaphragm     | **Options**  
                    | Used for analysis of a seismic requirements for IS 1893 (Indian) or ASCE 7 (US) codes.  
                    | Refer to [TR.28.2.1 Soft Story Checking](#) (on page 2529) for additional information.                                                                 |
| Set NL              | Opens the **Set NL** dialog, which is used to specify the maximum number of primary load cases for processing. It is required if multiple cycles of analysis is performed.  
                    | Type the **Maximum Number of Primary Loads** to provide in the input file and then click **OK**.  
                    | Refer to [TR.5 Set Command Specification](#) (on page 2413) for additional information.                                                                 |
| Set Echo            | Opens the **Set Echo** dialog, which is used to activate or deactivate the echoing of input commands to the output ( .ANL ) file. By default, Echo is On (i.e., input commands are echoed to the output file).  
                    | This option is used if you want to process any structure in which a node may be associated with more than 16 other joints.  
                    | Refer to [TR.5 Set Command Specification](#) (on page 2413) for additional information.                                                                 |
| Set Z Up            | By default, STAAD.Pro assumes that the global X and Z axes are horizontal and the Y axis is vertical. This command is used to orient the global Z axis as the vertical axis and sets X and Y to be the horizontal axes.  
                    | Refer to [TR.5 Set Command Specification](#) (on page 2413) for additional information.                                                                 |
| Set Displacement    | Opens the **Maximum allowable displacement tolerance** dialog, which is used to specify limiting values for nodal displacements for PDELTA and NONLINEAR analyses.  
                    | **Tip:** Make sure it is in the current length unit (see the bottom right hand corner of the STAAD.Pro window for the current input units).  
<pre><code>                | Refer to [TR.5 Set Command Specification](#) (on page 2413) for additional information.                                                                 |
</code></pre>
<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set Floor Load Tolerance</td>
<td>Opens the <strong>Tolerance used in Floor load calculation for length</strong> dialog, which is used to specify the tolerance for out of plane nodes in a floor load. The value is given as a length, in current units.</td>
</tr>
<tr>
<td>Set Floor Angle Tolerance</td>
<td>Opens the <strong>Tolerance used in Floor load calculation for angle</strong> dialog, which is used to specify the tolerance for out of plane nodes in a floor load. The value is given in degrees.</td>
</tr>
<tr>
<td>Set SDAMP</td>
<td>Opens the <strong>Damping ratio to be used for all springs</strong> dialog, which allows damping property of springs to be considered in computing the composite modal damping for each mode in a dynamic solution. Refer to [TR.5 Set Command Specification](on page 2413) for additional information.</td>
</tr>
<tr>
<td>Set Warp</td>
<td>Opens the <strong>Warping restraint ratio to compute torsional rigidity</strong> dialog, which allows end warping restraint to be considered for I-shaped members. Refer to [TR.5 Set Command Specification](on page 2413) for additional information. Assign a value which in the range 0 to 1. 0 indicates no restraint. 1 indicates full restraint. A fraction indicates a partial restraint.</td>
</tr>
<tr>
<td>Set Shear</td>
<td>This command is used for omitting the additional pure shear distortion stiffness terms in forming beam member stiffnesses. With this command you can exactly match simple textbook beam theory results. A message dialog opens to confirm you wish to omit the additional shear distortion terms. Refer to [TR.5 Set Command Specification](on page 2413) for additional information.</td>
</tr>
<tr>
<td>Set ITERLIM</td>
<td>Opens the Maximum number of tension/compression iterations dialog, which is used to override default limits for the maximum number of iterations of analysis performed for cases involving tension-only and compression-only components. Refer to [TR.5 Set Command Specification](on page 2413) for additional information.</td>
</tr>
<tr>
<td>Set NoWarning</td>
<td>This command will toggle off all warning messages in the output file. A message dialog opens to confirm you wish to toggle off all warning messages. Refer to [TR.5 Set Command Specification](on page 2413) for additional information.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| **Set Eigen Method** | Opens the **Set Eigen Method** dialog, which is used to specify the method used for eigen vectors or to use load-dependant Ritz vectors instead.  
Refer to [TR.5 Set Command Specification](on page 2413) for additional information.  
Refer to [G.17.3.1 Solution of the Eigenproblem](on page 2363) for details on load-dependent Ritz vectors. |
| **Cut Off Mode Shape** | Opens the **Cut Off Mode Shape** dialog, which is used to instruct the program to extract a higher or lower number of modes. By default, STAAD.Pro calculates only a specific number of modes during modal calculation, response spectrum analysis, and time history analysis. This defaults to the first six modes.  
**Option** | **Description**  
Allow Automatic Shifting | When the **Arnoldi/Lanczos** method is selected in the **Set Eigen Method** dialog, this option can be set to reduce memory demand. This allows for efficient solution of large models with a large number of degrees of freedom.  
Modes Per Shift | If the **Allow Automatic Shifting** option is set, then you must specify a targeted number of eigen modes to be searched in each shift.  
Initial Shift Frequency | When the **Arnoldi/Lanczos** method is selected in the **Set Eigen Method** dialog, you may specify an initial frequency shift value.  
Provide the maximum number of mode shapes we want the program to extract, and click **OK**.  
Refer to [TR.30.1 Cut-Off Frequency, Mode Shapes, or Time](on page 2539) for additional information. |
| **Cut Off Frequency** | Opens the **Cut Off Frequency** dialog, which is used to over-ride the built in cut off limit. During modal calculation, response spectrum analysis, and time history analysis, STAAD.Pro will extract only those modes whose frequency is below a built-in cut-off level.  
Refer to [TR.30.1 Cut-Off Frequency, Mode Shapes, or Time](on page 2539) for additional information. |
| **Cut Off Time** | Opens the **Cut Off Time** dialog, which is used to compute the results for a longer period of time than that at which the loads stop acting, specify the value using the **CUT OFF TIME** command. During time history analysis, by default, STAAD.Pro calculates displacements, forces, reactions and stresses for only up to a time which is equal to the duration of the longest acting forcing function or ground motion.  
Refer to [TR.30.1 Cut-Off Frequency, Mode Shapes, or Time](on page 2539) for additional information. |
Table 321: Analysis group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clear Above Commands</td>
<td>Opens the Delete Miscellaneous Commands dialog, which is to delete previously specified Miscellaneous commands such as Input Width, Set CO, etc. Check the boxes associated with the commands you want to delete and then click Delete.</td>
</tr>
<tr>
<td>Load List</td>
<td>Opens the Load List dialog, which is used to specify a list of existing load cases and load combination cases to be used for subsequent processes, like design, print, plot etc.</td>
</tr>
</tbody>
</table>

Table 322: Design Commands group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run Analysis</td>
<td>Performs the STAAD analysis as directed by input commands. The STAAD Analysis and Design window opens to display the progress of the design engine. When completed, you can select a next process and then click Done.</td>
</tr>
<tr>
<td>Steel Design</td>
<td>Opens the Steel Design - Whole Structure dialog, which is used to select a steel design code, select parameters to use for design, and assign parameters to the model.</td>
</tr>
<tr>
<td>Concrete Design</td>
<td>Opens the Concrete Design - Whole Structure dialog, which is used to select a concrete design code, select parameters to use for design, and assign parameters to the model.</td>
</tr>
<tr>
<td>Aluminium Design</td>
<td>Opens the Aluminum Design - Whole Structure dialog, which is used to select an aluminum design code, select parameters to use for design, and assign parameters to the model.</td>
</tr>
<tr>
<td>Timber Design</td>
<td>Opens the Timber Design - Whole Structure dialog, which is used to select a timber design code, select parameters to use for design, and assign parameters to the model.</td>
</tr>
</tbody>
</table>

Analysis - Whole Structure dialog

Used to view, add, and assign analysis commands to the STAAD input file.

Opens when the Analysis page is selected.

Command Tree  The dialog box shows a tree view (called Command Tree) of the commands in the Input Command File. The Command Tree is a pictorial view of the Input Command File with the commands displayed as "Nodes" of the tree in the same sequence in which they appear in the
Input Command File. Click on a command Node to expand the branch of the command and the command parameters appear as the "leaves" under that command node. The following paragraph describes the conventions followed in the Command Tree.

- Green Check (✓) marks: Commands which can be re-specified or modified.
- Gray Check (✗) marks: Commands which may not be changed.
- Question (?) marks: Commands which have been added but not assigned to any structural element (joint/member/element).

Define Commands... Opens the Analysis/Print Commands dialog (on page 3065), which is used to define the analysis commands to be included in the Input Command File.

Close Closes the dialog.

Help Opens the STAAD.Pro help window.

Analysis/Print Commands

Used to define the analysis commands to be included in the Input Command File. Commands are indicated by their respective tab.

Opens when the Analysis Commands tool is selected from the Analysis Data group on the Analysis and Design ribbon tab.

After current Select this option to add the analysis statement being defined in the Command Tree after the currently selected (highlighted) command.

Add Assigns the selected print command to the input command file.

Assign Used to assign the command to the current selection set.

Cancel Closes the dialog.

Help Opens the STAAD.Pro help window.

Perform Analysis tab

Used to direct STAAD.Pro to perform a linear elastic analysis.

Refer to TR.37.1 Linear Elastic Analysis (on page 2796) for additional information.

The dialog contains several print options, of which one may be selected.

No Print No analysis results will be printed to the output file.

Load Data Prints all the load data.

Statics Check Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

Statics Load Prints everything that Statics Check does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.
**Note:** Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

**Mode Shapes**
Prints mode shape values at the joints for all calculated mode shapes.

**Both**
This option is equivalent to the Load Data plus Statics Check options.

**All**
This option is equivalent to the Load Data plus Statics Check.

**Related Links**
- *A. To specify a linear elastic analysis* (on page 928)

**P Delta Analysis** tab

Used to direct STAAD.Pro to perform a second order analysis.

**Note:** This description of this feature is effective from STAAD.Pro 2007 Build 06 and greater.

Refer to TR.37.2 P-Delta Analysis Options (on page 2797) for additional information.

**Number of Iterations**
Specify the number of iterations to be performed. If specified, the program will perform the specified number of iterations whether or not the solution converges.

**Converge**
Select this option to check for convergence during analysis. Specify the maximum number of iterations used to check convergence, even if convergence has not been achieved. In this case, STAAD.Pro checks whether the solution is converging as successive iterations are performed. If the successive iterations result in divergent values of displacements for a particular load case, then the iterations are discontinued for the specific load case.

**Caution:** Do not use the Converge option when Number of Iterations is specified.

**Use Geometric Stiffness (Kg)**
P-Delta analysis including stress stiffening effect of the Kg matrix. With this option the global stiffness is adjusted with every iteration.

A regular STAAD.Pro P-Delta Analysis performs a first order linear analysis and obtains a set of joint forces from member/plates based on the large P-Delta effect. These forces are added to the original load vector. A second analysis is then performed on this updated load vector (5 to 10 iterations will usually be sufficient).

In the new P-Delta KG Analysis, that is, with the Kg option selected, the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is then performed using the original load vector. Large & small P-Delta effects are always included (1 or 2 iterations will usually be sufficient).

**Small Delta**
Select this option to include both P-small and large Delta effects (recommended), sometimes referred to as P-δ & P-Δ. Leave this box uncheck to only include the “PlargeDelta” effects (P-Δ only).

Without the Small Delta option (i.e., a default STAAD.Pro P-Delta analysis), STAAD.Pro performs a first order linear analysis and obtains a set of joint forces, from members/plates based on the large P-Delta effect, which are then added to the original load vector. A second analysis is then performed on this updated load vector.
With the **Small Delta** option selected, both the large & small P-Delta effects are included in calculating the end forces, (5 to 10 iterations will usually be sufficient).

**Print Option** Standard STAAD.Pro analysis print options:

- **No Print**  No analysis results will be printed to the output file.
- **Load Data**  Prints all the load data.
- **Statics Check**  Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
- **Statics Load**  Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.

**Note:** Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

- **Mode Shapes**  Prints mode shape values at the joints for all calculated mode shapes.
- **Both**  This option is equivalent to the **Load Data** plus **Statics Check** options.
- **All**  This option is equivalent to the **Load Data** plus **Statics Check**.

**Related Links**
- [G.17.2.1 P-Delta Analysis](#) (on page 2349)
- [A. To specify a P-Delta analysis](#) (on page 928)

**Perform Cable Analysis** tab

Used to assign the commands required to perform a non-linear cable analysis on the model. This requires the presence of non-linear cables in the structure.

**Note:**

Refer to [TR.37.3 Nonlinear Cable Analysis](#) (on page 2800) for additional information.

Refer to [G.8.2 Cable Members](#) (on page 2330) for an explanation of non-linear cable input options.

**Advanced Cable Analysis group**

The options in this group are used to specify the advanced cable analysis feature and options specific to that analysis type.

**Advanced Cable Analysis**

Set this option to specify an advanced cable analysis. Refer to [Section 1.18.2.8 of the Technical Reference Manual](#) (on page 2358) for details on the Advanced Cable analysis.

**Note:** Use of the Advanced Cable analysis feature requires the Advanced Analysis License. This option is the default if you have the license.
# Cable Analysis Parameters Group

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use Modified Newton-Raphson</td>
<td>Set this option to use the modified Newton-Raphson method, which can reduce computation effort in compared to the full Newton-Raphson method (at the expense of more iterations).</td>
</tr>
<tr>
<td>Use Geometric Matrix (Kg)</td>
<td>Set this option to specify the use of geometric stiffness.</td>
</tr>
<tr>
<td>Steps</td>
<td>Specify the number of load steps. The applied loads will be applied gradually in this many steps. Each step will be iterated to convergence. Should be in the range 5 to 145, with 145 as the recommended number.</td>
</tr>
<tr>
<td>Eq-Iterations</td>
<td>Maximum number of iterations permitted in each load step. Should be in the range of 10 to 500, with 300 as the recommended number.</td>
</tr>
<tr>
<td>Eq-tolerance</td>
<td>The convergence tolerance for the above iterations.</td>
</tr>
<tr>
<td>Sag Minimum</td>
<td>Cables (not trusses) may sag when tension is low. This is accounted for by reducing the E value. Sag minimum may be between 1.0 (no sag E reduction) and 0.0 (full sag E reduction). As soon as Sag Minimum becomes less than 0.95 the possibility exists that a converged solution will not be achieved without increasing the steps to 145 or the pretension loads. The Eq-Iterations may need to be 300 or more. The Eq-tolerance may need to be greater or smaller.</td>
</tr>
<tr>
<td>Stability Stiffness</td>
<td>Specify a stiffness matrix value that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for the first number of Load Steps. The amount added linearly decreases with each of the Load Steps. If a Stability Stiffness value is specified, it should be within the range of 0.0 to 1000.0. This parameter alters the stiffness of the structure.</td>
</tr>
<tr>
<td>Load Steps</td>
<td>The number of load steps over which the Stability Stiffness is gradually applied. Default is one step.</td>
</tr>
<tr>
<td>KSMALL</td>
<td>A stiffness matrix value that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for every load step. The range for KSMALL is between 0.0 and 1.0, with a default of 0.0. This parameter alters the stiffness of the structure.</td>
</tr>
</tbody>
</table>

### Print Option

Standard STAAD.Pro analysis print options:

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Print</td>
<td>No analysis results will be printed to the output file.</td>
</tr>
<tr>
<td>Load Data</td>
<td>Prints all the load data.</td>
</tr>
<tr>
<td>Statics Check</td>
<td>Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.</td>
</tr>
<tr>
<td>Statics Load</td>
<td>Prints everything that Statics Check does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.</td>
</tr>
</tbody>
</table>

**Note:** Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done...
for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

**Mode Shapes**
- Prints mode shape values at the joints for all calculated mode shapes.

**Both**
- This option is equivalent to the **Load Data** plus **Statics Check** options.

**All**
- This option is equivalent to the **Load Data** plus **Statics Check**.

**Related Links**
- *A. To specify a nonlinear cable analysis* (on page 931)
- *A. To specify a nonlinear cable analysis* (on page 931)

**Change** tab

Used to add the Change command to the input command file.

**Related Links**
- *G.19 Multiple Analyses* (on page 2401)
- *A. To add a change command* (on page 934)

**Perform Direct Analysis** tab

Used to instruct STAAD.Pro to perform a direct analysis per AISC 360 Appendix 7. This method accounts for the second order effects resulting from deformation in the structure due to applied loading, imperfections and reduced bending stiffness of members due to the presence of axial load.

This is a non-linear iterative analysis as the stiffness of the members is dependent upon the forces generated by the load. The analysis will iterate, in each step changing the member characteristics until the maximum change in any $\tau_b$ is less than the **Tau** tolerance. If the maximum change in any $\tau_b$ is less than 100x the **Tau** tolerance and the maximum change in any displacement degree of freedom is less than the **Displacement** tolerance; then the solution has converged for this case.

This analysis method is to be used in conjunction with Direct Analysis definition command. The Define Direct command should be added prior to instructing the program to perform this type of analysis.

Additionally, a direct analysis typically requires the definition of **Notional Loads** (on page 3029).

**Note:** Refer to *TR.37.5 Direct Analysis* (on page 2806) and *AD.2007-03.2.2 Direct Analysis* (on page 224) for additional information.

**Option**
- Select if the design methodology is Load Resistance and Factor (LRFD) or Allowable Stress (ASD).

**Tolerances**
- **Tau**
  - The limit to the change in the $\tau_b$ value in each iteration. If this value is exceeded by any member, then an additional iteration will be required unless the maximum number of iterations has been achieved

- **Displacement**
  - The limit to the change in the maximum displacement in each iteration. If this has been exceeded, then an additional iteration will be required unless the maximum number of iterations has been achieved.
**No. of Iterations**  
Specify the maximum number of iterations.

**PDelta Iterations**  
Select the number of iterations to be used for a PDelta with Small Delta analysis. Typically this should be from 5 to 25. The default number of 15 is recommended.

**Reduced EI**  
Check this option to use the reduced EI (Tau-b × 0.8 ×EI) for member section moment & section displacement. Clear the check box to use the full EI value.

**Perform Tau-b Iteration**  
Check this box to instruct the program to iterate Tau-b.

**Print Option**  
Standard STAAD.Pro analysis print options:

- **No Print**  
  No analysis results will be printed to the output file.

- **Load Data**  
  Prints all the load data.

- **Statics Check**  
  Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

- **Statics Load**  
  Prints everything that Statics Check does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.

  
  **Note:** Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

- **Mode Shapes**  
  Prints mode shape values at the joints for all calculated mode shapes.

- **Both**  
  This option is equivalent to the Load Data plus Statics Check options.

- **All**  
  This option is equivalent to the Load Data plus Statics Check.

**Related Links**
- [G.17.2.1.4 AISC 360 Direct Analysis](on page 2352)
- [A. To specify a direct analysis](on page 929)

**Generate Floor Spectrum** tab

Used to specify the calculation of floor and/or joint response spectrum when subjected to a time history loading. This information can be used in conjunction with equipment that will be supported by these floors and is often required by the equipment manufacturers.

**Note:** This feature requires STAAD.Pro V8i (release 20.07.04) or higher and an Advanced Analysis Package license.

The Floor Response Spectrum command must immediately follow an analysis command. That analysis can only contain a single time history load case. The groups of nodes (on page 3099) used to define a floor must be defined prior to adding this command. Results from this analysis will be displayed in the Floor Spectrum table in the Postprocessing workflow.

**Note:** Refer to [TR.37.10 Floor Spectrum Command](on page 2833) for additional information.
### Floor Groups table

Each row constitutes one floor which can have one or more floor groups assigned.

- **GX / GY / GZ** - Select the global direction options for which acceleration vs frequency spectrums will be generated for this floor.
- **Title** - Provide an optional title for this floor.
- **Node Groups** - Click the Selected Nodes to open a drop-down list of all previously defined node groups.

The resulting response spectra will be based on the collective responses of all the nodes in the selected groups.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Lowest Frequency</strong></td>
<td>Lowest frequency to be in the calculated spectrum. This value should be at least 0.01 Hz.</td>
</tr>
<tr>
<td><strong>Highest Frequency</strong></td>
<td>Highest frequency to be in the calculated spectrum.</td>
</tr>
<tr>
<td><strong>Frequency Interval</strong></td>
<td>The change in frequency values used for each interval calculation. The spectrum will be calculated at this interval from the <strong>Lowest Frequency</strong> to <strong>Highest Frequency</strong> specified.</td>
</tr>
<tr>
<td><strong>Damping Ratios table</strong></td>
<td>Up to ten damping values may be entered. One spectrum will be generated for each damping value for each global direction requested for each floor defined. The spectrum will be based on these modal damping ratios. Damping is specified as a fraction of one (that is, 3% damping should be entered as 0.03). The default is 0.05.</td>
</tr>
<tr>
<td><strong>Relative Acceleration?</strong></td>
<td>Select this option if there is ground motion defined and you want the spectrums to be based on the relative acceleration of the floor to the ground acceleration. Otherwise, absolute values are considered.</td>
</tr>
<tr>
<td><strong>Print Time History</strong></td>
<td>Select this option to print the time history acceleration being used in each spectrum calculation.</td>
</tr>
<tr>
<td><strong>Print Calculated Spectrum?</strong></td>
<td>Select this option to print the calculated spectrum.</td>
</tr>
</tbody>
</table>

### Related Links

- *A. To generate a floor spectrum* (on page 935)

### Nonlinear Analysis tab

Used to direct STAAD.Pro to perform a nonlinear analysis which can account for nonlinear effects of moderate displacement and small strain. This solution holds for where the element distortion is small and small rotations are assumed.

**Note:** This feature is available in STAAD.Pro V8i (release 20.07.05) and later; with the Displacement Limit controls requiring STAAD.Pro V8i (SELECTseries 2) release 20.07.07 or later.

**Note:** The Advanced Analysis Engine package is required to use this feature.

Refer to [TR.37.8 Geometric Nonlinear Analysis](on page 2830) for additional information.

The first analysis step must be stable, otherwise use ARC control to prevent instability. The procedure does not use follower loads. Loads are evaluated at the joints before the first step; then those loads translate with the joint but do not rotate with the joint. Equilibrium is computed in the displaced position.
Nonlinear entities such as tension/compression members, multilinear springs, gaps, etc. are not supported when using a nonlinear analysis. Additionally, nonlinear analysis does not account for post-buckling stiffness of members.

Note: The older NONLINEAR nn ANALYSIS command will adopt the new procedure unless a SET command is used. If a SET command is entered this will invoke the old procedure for backward compatibility.

The values for the Displacement Limit are used to limit analysis cycle to a specified displacement value and degree of freedom for a monitored node. When two or more Load Steps are specified, the calculated displacement or rotation is compared to the Target Value along or about the DOF for the specified Node each step. If the calculated displacement or rotation meets or exceeds the target value, then the analysis is stopped.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ARC</td>
<td>Specify a displacement control, which is the value of the absolute displacement limit for the first analysis step. If the maximum displacement is greater than this limit, ARC will calculate a new step size for the first step and a new value for Load Step. Value should be in current length units.</td>
</tr>
<tr>
<td>Note:</td>
<td>ARC = 0 indicates no displacement control.</td>
</tr>
<tr>
<td>Iterations</td>
<td>Max. Number of iterations to achieve equilibrium in the deformed position to the tolerance specified.</td>
</tr>
<tr>
<td>Tolerance</td>
<td>For convergence, two successive iteration results must have all displacements the same within this tolerance. Value entered is in current units.</td>
</tr>
<tr>
<td>Load Steps</td>
<td>Specify the number of load steps to be used. Load is applied in stages if entered. One means that all of the load is applied in the first step.</td>
</tr>
<tr>
<td>Rebuild Steps</td>
<td>Frequency of rebuilds of the tangent K matrix per load step &amp; iteration.</td>
</tr>
<tr>
<td></td>
<td>0 = once per load step</td>
</tr>
<tr>
<td></td>
<td>1 = every load step &amp; iteration</td>
</tr>
<tr>
<td>Use KG</td>
<td>Select this option to add the geometric stiffness, KG, to the stiffness matrix, K.</td>
</tr>
<tr>
<td>Node</td>
<td>Enter a node number or click [...] to enter the selected node to monitor during the geometric nonlinear analysis.</td>
</tr>
<tr>
<td>DOF</td>
<td>Select the degree of freedom to monitor during the analysis for specified Node.</td>
</tr>
<tr>
<td>Target Value</td>
<td>Specify a target displacement value along or about the selected DOF for the monitored Node.</td>
</tr>
<tr>
<td>Print Option</td>
<td>Standard STAAD.Pro analysis print options:</td>
</tr>
<tr>
<td>No Print</td>
<td>No analysis results will be printed to the output file.</td>
</tr>
<tr>
<td>Load Data</td>
<td>Prints all the load data.</td>
</tr>
<tr>
<td>Statics Check</td>
<td>Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.</td>
</tr>
<tr>
<td>Statics Load</td>
<td>Prints everything that Statics Check does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.</td>
</tr>
<tr>
<td>Note:</td>
<td>Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done</td>
</tr>
</tbody>
</table>

Note: Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done.
for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

**Mode Shapes**  
Prints mode shape values at the joints for all calculated mode shapes.

- **Both**  
  This option is equivalent to the **Load Data** plus **Statics Check** options.

- **All**  
  This option is equivalent to the **Load Data** plus **Statics Check**.

**Related Links**
- **G.17.2.3 Static Geometrically Nonlinear Analysis** (on page 2355)
- **A. To specify a nonlinear analysis** (on page 930)
- **A. To specify a nonlinear analysis** (on page 930)

**Perform Imperfection Analysis** tab

Used to instruct STAAD.Pro to perform an imperfection analysis, which will reflect the secondary effects only if the camber and/or drift is specified in a using the Define Imperfection command. For combination of load cases, with imperfection, use the Repeat Load specification rather than the Load Combination.

Refer to sections **TR.26.6 Member Imperfection Information** (on page 2511) and **TR.37.9 Imperfection Analysis** (on page 2832) for additional details.

There are no other parameters associated with this command.

**Print Option**  
Standard STAAD.Pro analysis print options:

- **No Print**  
  No analysis results will be printed to the output file.

- **Load Data**  
  Prints all the load data.

- **Statics Check**  
  Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

- **Statics Load**  
  Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.

  **Note:** Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

- **Mode Shapes**  
  Prints mode shape values at the joints for all calculated mode shapes.

- **Both**  
  This option is equivalent to the **Load Data** plus **Statics Check** options.

- **All**  
  This option is equivalent to the **Load Data** plus **Statics Check**.

**Related Links**
- **G.17.2.4 Imperfection Analysis** (on page 2355)
- **A. To specify an imperfection analysis** (on page 932)
- **M. To assign member imperfection for members** (on page 800)
Perform Buckling Analysis tab

Used to instruct STAAD.Pro to perform a buckling analysis.

Refer to TR.37.4 Buckling Analysis (on page 2804) for additional information.

Iterative method

When this option is selected, the program will perform a P-Delta analysis including Kg Stiffening (geometric stiffness of members & plates) due to large & small P-Delta effects. If a non-singular stiffness matrix can be created, then buckling has not occurred. Then the load is increased from the last increment repeatedly until buckling does occur. Then the load is decreased halfway back to the prior increment. This bounds the buckling factor between the last 2 increments. Then STAAD.Pro proceeds to halve the interval until either the change between increments is 0.1% of each other, or the specified number of increments has been exceeded. The resulting factor is reported in the output file. The buckling deformed shape is simply the deformed shape from a static analysis with the near buckling load applied. This could appear more like a crushing, small displacement shape rather than a buckling mode shape. At least 15 iterations are recommended. Buckling will be applied to all primary cases.

Number of Iterations

Maximum number of iterations. Fifteen is recommended.

Note: This parameter is only used for the iterative method. It is ignored for the eigen method.

Eigen method

This buckling method may only be used if an Advanced Analysis license is available. When using the Advanced Solver, the corresponding “buckling modes” are included in the output file.

Print Option

Standard STAAD.Pro analysis print options:

- No Print: No analysis results will be printed to the output file.
- Load Data: Prints all the load data.
- Statics Check: Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
- Statics Load: Prints everything that Statics Check does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.

Note: Since PRINT STATIC LOAD generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

- Mode Shapes: Prints mode shape values at the joints for all calculated mode shapes.
- Both: This option is equivalent to the Load Data plus Statics Check options.
- All: This option is equivalent to the Load Data plus Statics Check.

Related Links

- G.17.2.2 Buckling Analysis (on page 2353)
- A. To specify buckling analysis (on page 932)
Perform Pushover Analysis tab

Used to instruct the program to perform a pushover analysis. This command has no other parameters.

Related Links

- TR.37.7 Pushover Analysis (on page 2819)
- G.17.4 Pushover Analysis (on page 2376)
- M. Pushover Loads (on page 861)
- A. To specify a pushover analysis (on page 933)

Analysis/Print Commands dialog (Pre Print)

Used to define the pre-analysis print commands to be included in the Input Command File. Commands are indicated by their respective tab.

Opens when:

- The Pre Analysis Commands tool is selected from the Analysis Data group on the Analysis and Design ribbon tab, or
- Define Commands is clicked on the Pre Analysis Print dialog

Problem Statistics tab

This command was previously used to include structure information such as total number of joints, members, supports, disk space requirements, the estimated band-width of the stiffness matrix, etc.

Note: This command has been deprecated and will be ignored by the engine during analysis.

Joint Coordinates tab

Used to add the joint coordinate values.

Member Information tab

Used to add member length, member incidences, beta angles, and member specifications such as truss member, and the member release conditions at start and end of the member (i.e., 1 = released, 0 = not released).

Material Properties tab

Used to add material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members.

Note: This command is available for members only.

Support Information tab

Used to add information regarding restraints, releases and spring constant values of the supports and the joint numbers where they are located.
All tab
Used to print joint coordinates, member information, member properties, material properties, and support information, in that order.

Element Information tab
Used to add element incidences, element thickness, and Poisson ratios for plate elements.

Solid Information tab
Used to add element incidences and Poisson ratios for solid elements.

Member Properties tab
Used to add member properties including cross sectional area, moments of inertia, and section moduli in both axes.

After current
Select this option to add the pre analysis print statement being defined in the Command Tree after the currently selected (highlighted) command.

Add
Assigns the selected print command to the input command file.

Assign
Used to assign the command to the current selection set.

Cancel
Closes the dialog.

Help
Opens the STAAD.Pro help window.

Related Links
• A. To specify pre-analysis commands (on page 936)

Analysis/Print Commands dialog (Post Print)

Used to define the post-analysis print commands to be included in the Input Command File. Commands are indicated by their respective tab.

Opens when:

Load List tab
Used to select the Load Cases to be included in the output.

Load Cases list
All load cases and combinations in the input file are listed here.

Selection tools

Load List
Contains all the selected load cases which will be included in the output.

Section tab
Used to specify up to five sections along the length of frame members where forces and moment results are desired.
Joint Displacement tab
Used to print joint displacements in a tabulated form. The displacements for all six degrees of freedom will be printed for all specified joints for all specified load cases.

Member Forces tab
Used to print member forces (i.e. axial force (AXIAL), shear force in local Y and Z axes (SHEAR-Y and SHEAR-Z), torsional moment (TORSION), moments about local Y and Z axes (MOM-Y and MOM-Z) in a tabulated form for all specified members for all specified load cases.

Global Selection this option to display the member forces in terms of the Global axis. Otherwise, member forces are presented in local member axis.

Support Reactions tab
Used to print support reactions in a tabulated form for all specified supports for all specified load cases.

Story Drift tab
Used to obtain a print-out of the average lateral displacement of all joints at each vertical level of the structure.

Cg tab
Prints the coordinates of the center of gravity of the structure.

Surface Forces tab

Physical Member Forces tab
Used to print produce member forces for all the analytical members in the physical member (PMEMBER) group

Buckling Shapes tab
Used to print the buckling shape for a Buckling Analysis.

Dia CR tab
Used to print the center of rigidity and center of mass for each rigid floor diaphragm in the model, which are used to determine the natural torsion moment for seismic loads which use that option.

Cable Sag
Used to include the sagging deflected shape of cable members for an Advanced Cable Analysis.

The output of a successful advanced cable analysis will report the cable sag in local XYZ coordinates. Post-analysis print will calculate the actual, nonlinear cable displacements along the length of cable.

Mode Shapes tab
Used to print mode shape values at every joint for all calculated modes.
Element Stress Solid tab
Used to print stresses at the center of solid elements for all specified solids for all specified load cases.

- **Print Element Stresses**
  Select this option to print center stresses.

- **Print Element Joint Stresses**
  Select this option to print stresses at the corner nodes.

Section Displacement tab
Used to calculate and print section displacements at sections (intermediate points) of frame members.

- **Number of Sections**
  Number of intermediate sections at which results are to be reported.

- **Options**
  The *Maximum* option prints only the maximum local displacements from among all load cases. The *Save* option saves the section displacements to a file for future reference. The *No Print* option does not print the section displacements to the .ANL file and should be used in conjunction with the *Save* option.

- **Assign**
  Select if the command is to be assigned to all members in the current view or to a selection of members.

**Note**: Selection must be made prior to selecting the menu command item.

Force tab
Used to print force/moment envelopes for frame members for the specified *Number of Sections*. It prints the maximum and minimum values for every section for each specified load case.

- **Number of Sections**
  Specify the number of equally-spaced sections to be considered.

Max Force tab
Used to print force/moment envelopes for frame members for the specified *Number of Sections*. It prints the maximum and minimum values of all sections from among the specified load cases.

- **Number of Sections**
  Specify the number of equally-spaced sections to be considered.

Analysis Results tab
Used to print Joint Displacements, Support Reactions, and Member Forces for all specified joints/members for all specified load cases. A message dialog will open to confirm you wish to add the Analysis Results command.

Member Section Forces tab

Member Stresses tab
Used to print member stresses at the start and end joints and at all specified intermediate sections of selected members. These stresses include axial, (axial force over area), bending-y (moment-y over section modulus in
local y-axis), bending-z (moment-z over section modulus in local z-axis), shear stress in both local y and z
directions and combined (absolute combination of axial, bending-y and bending-z) stresses.

**Element Forces/Stresses tab**

Used to print stresses (FX, FY, FXY, QX, QY), moments per unit width (MX, MY, MXY) and principal stresses
(SMAX, SMIN, TMAX) at the centroid of plate elements. The Von Mises stresses (VONT, VONB) as well as the
angle (ANGLE) defining the orientation of the principal planes are also printed.

Selecting the Print Nodal Point Forces & Moments checkbox will also print stresses and moments at the corner
nodes. The Print Force at Point option can be used to print element forces at any point within the element.

**Note:** For further information, please refer to G.5.1 Plate and Shell Elements (on page 2308) and TR.41 Section
Specification (on page 2839).

**Related Links**

- A. To specify post-analysis print commands (on page 939)

**Floor Diaphragm Options** dialog

Used to select a code by which the seismic code checks should be performed.

Opens when the Miscellaneous Commands > Floor Diaphragm Options tool is selected in the Analysis Data
group on the Analysis and Design ribbon tab.

**Design Code**

Select one of the following codes:

- IS1893 2002
- ASCE7-05
- IS1893 2016

**Check Soft Story**

Check this option to check for soft story (vertical irregularities) between two adjacent
diaphragms.

**Check Irregularities** (IS1893 2016 only) Check this option to check for plan as described in TR.28.2.2 Check
Irregularities (on page 2531). This will check for torsion irregularities, reentrant corner
irregularities, mass irregularities, and irregular modes of oscillation (when zone 4 or 5
have been selected in the static seismic definition).

**Base Level**

If the base level of the structure is not at the minimum global Y value of the entire
model, then check this option and type the value in the current length units.
No of Diaphragms  Displays the number of diaphragms in the model. This is field is for information only and cannot be edited from this dialog.

Related Links
- *A. To check for soft stories and seismic code irregularities* (on page 937)
- **TR.31.2.14 IBC 2015 Seismic Load Definition**
- *A. To check for soft stories and seismic code irregularities* (on page 937)
- **TR.31.2.14 IBC 2015 Seismic Load Definition** (on page 2600)
- *A. To check for soft stories and seismic code irregularities* (on page 937)

**Load List** dialog

Used to specify a list of existing load cases and load combination cases to be used for subsequent processes, like design, print, plot etc.

Opens when the **Load List** tool is selected from the **Analysis Data** group on the **Analysis and Design** ribbon tab.

**Load Cases list**  All existing load cases including combination load cases are listed in the left hand side list box. These load cases can be selected and transferred to the right hand side **Load list**.

**List Operators**

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected primary load case to the Load list</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all primary load cases to the Load list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all entries in the Selected Load Combinations list, load combination is placed in the Load Cases list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected entry from the Selected Load Combinations list.</td>
</tr>
</tbody>
</table>

**Load list**  This list box contains the list of selected load cases to be included in the load group.

Related Links
- *A. To create a load list* (on page 937)

**STAAD Analysis and Design** dialog

This dialog box displays the status of the analysis process.

If an error occurs during the analysis, the above dialog box displays the error message.
For example, if you are running a demonstration version, or the hardware lock is not found, you may encounter an error message like the following:
DEMO VERSION: Analysis Limited to 6 members.

The following options allow you to control what happens when you click Done.

- **View Output File** – opens the STAAD Output Viewer with the analysis results presented in a textual format.
- **Go to Post Processing Mode** – opens the STAAD.Pro Postprocessor workflow where you can graphically view results.
- **Stay in Modeling Mode** – closes the dialog while staying in the Analytical Modeling workflow.

**Related Links**
- *STAAD Analysis and Design dialog* (on page 3080)
- *A. To perform an analysis in STAAD.Pro* (on page 942)
- *STAAD Analysis and Design dialog* (on page 3080)
- *A. To perform an analysis in STAAD.Pro* (on page 942)

**Steel Design - Whole Structure** dialog

Used to select a steel design code, select parameters to use for design, and assign parameters to the model.

Opens when the Design > Steel Design tool is selected from the Design group on the Analysis and Design ribbon tab.

**Note:** For additional information on steel design codes available in STAAD.Pro, refer to D. Available Steel Design Codes (on page 944). The D. Design Codes (on page 1366) section contains detailed descriptions of the design parameters used for each code.

<table>
<thead>
<tr>
<th><strong>Current Code</strong></th>
<th>Select the code to use for design from the drop-down list.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Command Tree</strong></td>
<td>Displays a summary of all input file commands. Each command is represented with that commands parameters as child elements. Click the [+] icon to expand the command to display parameters.</td>
</tr>
<tr>
<td></td>
<td>• Green Check (✔️) marks: Commands which can be re-specified or modified.</td>
</tr>
<tr>
<td></td>
<td>• Gray Check (✔️) marks: Commands which may not be changed.</td>
</tr>
<tr>
<td></td>
<td>• Question (? ) marks: Commands which have been added but not assigned to any structural element (joint/member/element).</td>
</tr>
<tr>
<td><strong>Highlight</strong></td>
<td>Select this option to have model objects highlighted in the View Window as commands are selected in the Command Tree</td>
</tr>
<tr>
<td><strong>Assigned</strong></td>
<td>Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the &quot;Use Cursor to assign&quot; method, the following happens.</td>
</tr>
<tr>
<td><strong>Geometry</strong></td>
<td>• Click on the member or element once - the attribute gets assigned.</td>
</tr>
<tr>
<td><strong>Toggle Assign</strong></td>
<td>• Click on the same member or element a second time - the attribute gets de-assigned.</td>
</tr>
</tbody>
</table>
Click on the same member or element again, - the attribute gets re-assigned. Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.

Select Parameters… Opens the **Parameters Selection dialog** (on page 3082), which is used to select the parameters included in the STAAD.Pro input file for design.

Define Parameters… Opens the **Design Parameters dialog** (on page 3083) for the selected code, which is used to define design parameters and add them to the input file.

Commands… Opens the **Design Commands dialog** (on page 3084), which is used to add design commands for the selected design code to the input file.

Assignment Method Select the method for command/parameter assignment.

- **Assign to Selected <objects>** - Available once objects are selected in the View window. Used to assign a Design option to selected joints, members or elements, first select the command from the **Command Tree**. Note that the label for this button changes depending on whether nodes, members or elements are selected.

- **Assign to View** - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams).

- **Use Cursor To Assign** - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The **Toggle Assign** option can also be used to aid in this mode.

- **Assign To Edit List** - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button.

- **Select Group/Deck** - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the **Select Group/Deck dialog** (on page 3084), which is used to select the group for assignment.

Assign Used to assign the selected parameter to the selected objects.

Close Closes the dialog.

Help Opens the STAAD.Pro help window.

Related Links
- **D. To specify steel design code and parameters** (on page 951)
- **TR.48 Steel and Aluminum Design Specifications** (on page 2850)
- **D. To design steel members in groups** (on page 951)
- **D. To specify steel design commands** (on page 952)

**Parameters Selection dialog**

Used to select the parameters included in the **Design Parameters** dialog.

Opens when **Select Parameters** is clicked on a **<material> Design - Whole Structure** dialog.
Available Parameters
Lists all available parameters for the current design code.

Selected Parameters
Parameters added to this list will be made available for use for the current design code in the Define Parameters... dialog.

**Note:** By default, all parameters are added to the list.

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Include the selected design parameter in the Selected Parameters list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Include all design parameters in the Selected Parameters list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all design parameters from the Selected Parameters list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected design parameter from the Selected Parameters list.</td>
</tr>
</tbody>
</table>

**OK**
Updates the Parameters list in the Design Parameters dialog (on page 3083) and closes the Parameter Selection dialog.

**Cancel**
Closes the dialog without updating changing the selected parameters.

**Help**
Opens the STAAD.Pro help window.

### Design Parameters dialog
Used to define design parameters for the current design code and add them to the input file.

Opens when Define Parameters is clicked on a <material> Design - Whole Structure dialog.

**Parameter list**
All parameters available for the current design code are displayed here.

**Note:** For details on each design code parameters, refer to D. Design Codes (on page 1366).

**After Current**
Select this option to have the new design parameter inserted after the currently selected in the Design dialog explorer list.

**Add**
Adds the selected design parameter to the current code in the input file. The design parameter must then be assigned to model objects.

**Assign**
Active if appropriate model objects were selected prior to clicking Define Parameter. Adds the design parameter to the input file and assigns the design parameter to the selected objects.

**Close**
Closes the dialog.

**Help**
Opens the STAAD.Pro help window.

**Related Links**
- D. To specify steel design code and parameters (on page 951)
Design Commands dialog

Used to add design commands for the selected design code to the input file.

Opens when Commands is clicked on a <material> Design - Whole Structure dialog.

Command list
All design commands available for the current design code are displayed here.

Note: For details on the available design commands for each material and code, refer to D. Design Codes (on page 1366).

After Current
Select this option to have the new design command inserted after the currently selected command in the Design dialog explorer list.

Add
Adds the selected design command to the current code in the input file. The design command must then be assigned to model objects.

Assign
Active if appropriate model objects were selected prior to clicking Commands. Adds the design command to the input file and assigns the design command to the selected objects.

Close
Closes the dialog.

Help
Opens the STAAD.Pro help window.

Related Links

- TR.48 Steel and Aluminum Design Specifications (on page 2850)
- D. To design steel members in groups (on page 951)
- D. To specify steel design commands (on page 952)
- D. To specify concrete beam design command (on page 1003)
- D. To generate concrete take off (on page 1004)
- D. To specify aluminum design commands (on page 1363)
- D. To generate aluminum take off (on page 1364)
- D. To generate steel take off (on page 953)
- D. To specify timber design commands (on page 1365)

Select Group / Deck

Used to select the group for assignment of commands.

Opens when Select Group/Deck is clicked in a <material> Design - Whole Structure dialog.

List
Lists all previously defined groups and composite decks in the input file, organized by type.

OK
Assigns the parameter selected in the Design dialog list to the selected group or deck.

Cancel
Closes the dialog without making any assignments.
Concrete Design - Whole Structure dialog

Used to select a concrete design code, select parameters to use for design, and assign parameters to the model.

Opens when the Design > Concrete Design tool is selected from the Design group on the Analysis and Design ribbon tab.

**Note:** For additional information on concrete design codes available in STAAD.Pro, refer to D. Available Concrete Design Codes (on page 1000). The D. Design Codes (on page 1366) section contains detailed descriptions of the design parameters used for each code.

| Current Code | Select the code to use for design from the drop-down list. |
| Command Tree | Displays a summary of all input file commands. Each command is represented with that commands parameters as child elements. Click the [+] icon to expand the command to display parameters. |
| | • Green Check (✓) marks: Commands which can be re-specified or modified. |
| | • Gray Check (✓) marks: Commands which may not be changed. |
| | • Question (?) marks: Commands which have been added but not assigned to any structural element (joint/member/element). |
| Highlight Assigned Geometry | Select this option to have model objects highlighted in the View Window as commands are selected in the Command Tree |
| Toggle Assign | Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens. |
| | • Click on the member or element once - the attribute gets assigned. |
| | • Click on the same member or element a second time - the attribute gets de-assigned. |
| | • Click on the same member or element again, - the attribute gets re-assigned. |
| | Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned. |
| Select Parameters... | Opens the Parameters Selection dialog (on page 3082), which is used to select the parameters included in the STAAD.Pro input file for design |
| Define Parameters... | Opens the Design Parameters dialog (on page 3083) for the selected code, which is used to define design parameters and add them to the input file |
| Commands... | Opens the Design Commands dialog (on page 3084), which is used to add design commands for the selected design code to the input file. |
| Assignment Method | Select the method for command/parameter assignment. |
| | • Assign to Selected <objects> - Available once objects are selected in the View window. Used to assign a Design option to selected joints, members or elements, first select the command from the Command Tree. Note that the label for this button changes depending on whether nodes, members or elements are selected. |
Assignment Methods:

- **Assign to View** - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams).
- **Use Cursor To Assign** - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The **Toggle Assign** option can also be used to aid in this mode.
- **Assign To Edit List** - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button.
- **Select Group/Deck** - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the **Select Group/Deck dialog** (on page 3084), which is used to select the group for assignment.

Once an assignment method has been selected, the dialog:

- **Assign** - Used to assign the selected parameter to the selected objects.
- **Close** - Closes the dialog.
- **Help** - Opens the STAAD.Pro help window.

Related Links:

- D. To specify concrete design code and parameters (on page 1002)
- D. To specify concrete beam design command (on page 1003)
- D. To generate concrete take off (on page 1004)

## Aluminum Design - Whole Structure dialog

Used to select an aluminum design code, select parameters to use for design, and assign parameters to the model.

Opens when the **Design > Aluminum Design** tool is selected from the **Design** group on the **Analysis and Design** ribbon tab.

**Note:** For additional information on aluminum design codes available in STAAD.Pro, refer to D. Available Aluminum Design Codes (on page 1362). The D. Design Codes (on page 1366) section contains detailed descriptions of the design parameters used for each code.

**Current Code**

Select the code to use for design from the drop-down list.

**Command Tree**

Displays a summary of all input file commands. Each command is represented with that command's parameters as child elements. Click the [+/-] icon to expand the command to display parameters.

- **Green Check (✓)** marks: Commands which can be re-specified or modified.
- **Gray Check (✓)** marks: Commands which may not be changed.
- **Question (❓)** marks: Commands which have been added but not assigned to any structural element (joint/member/element).
Highlight Assigned Geometry

Toggle Assign

Select this option to have model objects highlighted in the View Window as commands are selected in the Command Tree.

Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens:

- Click on the member or element once - the attribute gets assigned.
- Click on the same member or element a second time - the attribute gets de-assigned.
- Click on the same member or element again, - the attribute gets re-assigned.

Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.

Select Parameters…

Opens the Parameters Selection dialog (on page 3082), which is used to select the parameters included in the STAAD.Pro input file for design.

Define Parameters…

Opens the Design Parameters dialog (on page 3083) for the selected code, which is used define design parameters and add them to the input file.

Commands…

Opens the Design Commands dialog (on page 3084), which is used to add design commands for the selected design code to the input file.

Assignment Method

Select the method for command/parameter assignment.

- Assign to Selected <objects> - Available once objects are selected in the View window. Used to assign a Design option to selected joints, members or elements, first select the command from the Command Tree. Note that the label for this button changes depending on whether nodes, members or elements are selected.
- Assign to View - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams).
- Use Cursor To Assign - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The Toggle Assign option can also be used to aid in this mode.
- Assign To Edit List - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button.
- Select Group/Deck - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the Select Group/Deck dialog (on page 3084), which is used to select the group for assignment.

Once an assignment method has been selected, the

Assign

Used to assign the selected parameter to the selected objects.

Close

Closes the dialog.

Help

Opens the STAAD.Pro help window.

Related Links

- D. To specify aluminum design code and parameters (on page 1363)
- D. To specify aluminum design commands (on page 1363)
**Timber Design - Whole Structure** dialog

Used to select a timber design code, select parameters to use for design, and assign parameters to the model.

Opens when the **Design > Timber Design** tool is selected from the **Design** group on the **Analysis and Design** ribbon tab.

**Note:** For additional information on timber design codes available in STAAD.Pro, refer to D. **Available Timber Design Codes** (on page 1364). The D. **Design Codes** (on page 1366) section contains detailed descriptions of the design parameters used for each code.

<table>
<thead>
<tr>
<th>Tab</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Current Code</td>
<td>Select the code to use for design from the drop-down list.</td>
</tr>
<tr>
<td>Command Tree</td>
<td>Displays a summary of all input file commands. Each command is represented with that commands parameters as child elements. Click the [+] icon to expand the command to display parameters.</td>
</tr>
<tr>
<td></td>
<td>- Green Check (✓) marks: Commands which can be re-specified or modified.</td>
</tr>
<tr>
<td></td>
<td>- Gray Check (✗) marks: Commands which may not be changed.</td>
</tr>
<tr>
<td></td>
<td>- Question (❓) marks: Commands which have been added but not assigned to any structural element (joint/member/element).</td>
</tr>
<tr>
<td>Highlight Assigned Geometry</td>
<td>Select this option to have model objects highlighted in the View Window as commands are selected in the <strong>Command Tree</strong></td>
</tr>
<tr>
<td>Toggle Assign</td>
<td>Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the &quot;Use Cursor to assign&quot; method, the following happens.</td>
</tr>
<tr>
<td></td>
<td>- Click on the member or element once - the attribute gets assigned.</td>
</tr>
<tr>
<td></td>
<td>- Click on the same member or element a second time - the attribute gets de-assigned.</td>
</tr>
<tr>
<td></td>
<td>- Click on the same member or element again, - the attribute gets re-assigned.</td>
</tr>
<tr>
<td></td>
<td>Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.</td>
</tr>
<tr>
<td>Select Parameters...</td>
<td>Opens the <strong>Parameters Selection dialog</strong> (on page 3082), which is used to select the parameters included in the STAAD.Pro input file for design</td>
</tr>
<tr>
<td>Define Parameters...</td>
<td>Opens the <strong>Design Parameters dialog</strong> (on page 3083) for the selected code, which is used define design parameters and add them to the input file</td>
</tr>
<tr>
<td>Commands...</td>
<td>Opens the <strong>Design Commands dialog</strong> (on page 3084), which is used to add design commands for the selected design code to the input file.</td>
</tr>
<tr>
<td>Assignment Method</td>
<td>Select the method for command/parameter assignment.</td>
</tr>
<tr>
<td></td>
<td>- Assign to Selected &lt;objects&gt; - Available once objects are selected in the View window.</td>
</tr>
<tr>
<td></td>
<td>Used to assign a Design option to selected joints, members or elements, first select the command from the <strong>Command Tree</strong>. Note that the label for this button changes depending on whether nodes, members or elements are selected.</td>
</tr>
</tbody>
</table>
• Assign to View - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams).

• Use Cursor To Assign - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The Toggle Assign option can also be used to aid in this mode.

• Assign To Edit List - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button.

• Select Group/Deck - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the Select Group/Deck dialog (on page 3084), which is used to select the group for assignment.

Once an assignment method has been selected, the

Assign Used to assign the selected parameter to the selected objects.
Close Closes the dialog.
Help Opens the STAAD.Pro help window.

Related Links
• TR.52 Timber Design Specifications (on page 2857)
• D. To specify timber design code and parameters (on page 1365)
• D. To specify timber design commands (on page 1365)

Utilities tab

Table 323: Geometry Tools group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structure Tools &gt;</td>
<td></td>
</tr>
<tr>
<td>Check Multiple Structures</td>
<td>Opens the List of Structures dialog, which is used to determine if the current model consists of more than one unconnected structure. Select a Structure in the list to see each structure highlighted.</td>
</tr>
<tr>
<td>Check Beam Plate Connectivity</td>
<td>Used to check for plates that are improperly connected to beams. Beams must be connected to plates at their nodes in order to ensure proper coupling and load transfer. This tool will inform the user if any improper connections are present in the model.</td>
</tr>
</tbody>
</table>
### Tool Name | Description
--- | ---
**Merge Properties** | Used to merge the properties of two or more similar objects. When a STAAD input file has a number of references of the same property, this tool can be used to consolidate all these properties into a single command. When a STAAD file has a number of references of the same property, there is now a tool to consolidate all these properties into a single command. Clicking the Yes button, all instances of a given section property will be collated into a single property reference. **Note:** Properties references with differing additional parameters will not be collated. Properties references with differing assigned material properties will not be collated.

**Cut Section** | Opens the Section dialog, which is used to cut a section through the structure along a specified global plane at a desired location of the 3rd axis.

**Check Duplicate Nodes** | Opens the Remove Duplicate Nodes dialog, which is used to set the tolerance distance between two nodes the program should consider as duplicate. Used for detecting the presence of two or more instances of the same node.

**Highlight Orphan Nodes** | Highlights all nodes in the structure which are not connected to any member, element, or solid.

**Remove Orphan Nodes** | Used to remove all detected orphan nodes.

**Node to Node Distance** | Display the distance between nodes.

**Remove Node to Node Distance** | Used to remove the display of all node to node dimensions from the current view.

**Check Duplicate Beams** | Opens the Remove Duplicate Beams dialog, which is used to set the tolerance distance between two beams the program should consider as duplicate. Used for detecting the presence of two or more instances of the same beam.
<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Check Zero Length Members</td>
<td>Opens the <strong>Zero Length Tolerance</strong> dialog, which is used to set the tolerance for zero length members. A member connected between duplicate nodes, which have the same (X,Y,Z) coordinates, will have a length of zero. This tool detects such members.</td>
</tr>
<tr>
<td>Check Overlapping Collinear Members</td>
<td>When two members are collinear, and further, at least one of the nodes of one of those members happens to lie within the span of the other, but the two members are not connected at that node, those two members are considered as overlapping collinear members. This tool detects such members. The usefulness of this tool comes from the fact that it enables you to detect modeling errors which are not easily visible. Two lines overlapping on the drawing area become indistinguishable on large models, and this is one of the tools that can spot such errors.</td>
</tr>
<tr>
<td>Redefine Incidence</td>
<td>Used to reverse the incidence of selected beams so that the global coordinates of the end node are farther from the origin than that of the start node.</td>
</tr>
<tr>
<td>Dimension Beams</td>
<td>Display the dimension of the members in the structure.</td>
</tr>
<tr>
<td>Check Duplicate Plates</td>
<td>Opens the <strong>Remove Duplicate Plates</strong> dialog, which is used to set the tolerance distance between two plates the program should consider as duplicate. Used for detecting the presence of two or more instances of the same plate.</td>
</tr>
<tr>
<td>Check for Warped Plates</td>
<td>A warped plate is defined as a four-noded plate whose nodes do not lie on the same plane. This tool detects such plates.</td>
</tr>
<tr>
<td>Check Improperly Connected Plates</td>
<td>Checks the model for plates that overlap or intersect each other at their boundaries and, if found, opens the Overlapping Collinear Plates dialog. Typically, these overlaps cause improper load transfers or instabilities in the model.</td>
</tr>
<tr>
<td>Check for Solids with Negative Volume</td>
<td>Used to verify that the solid elements in their model have the proper sequence (order) of node numbering to prevent warnings in the output file of solids containing negative volumes.</td>
</tr>
<tr>
<td>Check for Warped Solids</td>
<td>Used to check if a solid element is warped. Warped solids cannot be analyzed and will produce errors in the output file.</td>
</tr>
</tbody>
</table>
### Tool Name

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Physical Member Restraints</strong></td>
<td>Used to automatically generate top and bottom flange restraint conditions for the selected physical members.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> The <code>PBRACE</code> command generated by this menu item is valid only for Member Design (on page 1649) per AS 4100-1998 (Australian) steel design.</td>
</tr>
<tr>
<td></td>
<td>When selected, the program will search through all the nodes along the Physical Member and determine if any other beam is connected besides the beams in the current physical member definition. If another beam is present, a full restraint is placed on both the flanges. Otherwise, an unrestrained condition is imposed on both the flanges.</td>
</tr>
<tr>
<td><strong>Groups</strong></td>
<td>Opens the Create Group dialog, which is used to cluster a set of joints, beams, plates or solids into a single entity identified by a distinct name.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If no groups exist, you will be prompted to create a group using the Define Group Name dialog.</td>
</tr>
</tbody>
</table>

#### Table 324: Physical Model group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Drop Physical Model</strong></td>
<td>For models that have a linked physical model, this will open a confirmation dialog asking if you want to remove that link. Removing this link will enable the model to be edited using all analytical modeling tools as well as remove the disabled sections of the input file in the STAAD.Pro Editor.</td>
</tr>
</tbody>
</table>

#### Table 325: Query group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Beams</strong></td>
<td>Used to view detailed information for one beam at a time in a dialog.</td>
</tr>
<tr>
<td><strong>Physical Members</strong></td>
<td>Used to view detailed information for one physical member at a time in a dialog.</td>
</tr>
<tr>
<td><strong>Plates</strong></td>
<td>Used to view detailed information for one plate at a time in a dialog.</td>
</tr>
<tr>
<td><strong>Solids</strong></td>
<td>Used to view detailed information for one solid at a time in a dialog.</td>
</tr>
</tbody>
</table>
### Tool Name and Description

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes</td>
<td>Used to view detailed information for one node at a time in a dialog.</td>
</tr>
</tbody>
</table>

### Table 326: Display group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Insert Text</td>
<td>Opens the <strong>Write Text to Insert</strong> dialog, which is used to add custom text to the View window (and, subsequently to pictures you take of the window).</td>
</tr>
</tbody>
</table>

### Table 327: Edit group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command File</td>
<td>Opens the current input command file (file extension <code>.std</code>) in the STAAD.Pro Editor. If any change has been made in the structure that has not been saved, you are prompted to save the structure first. <strong>Note:</strong> Refer to the [STAAD.Pro Editor help](on page 2251) for additional assistance.</td>
</tr>
</tbody>
</table>

### Table 328: View group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis Output</td>
<td>Opens the results of a successful Analysis and Design run in the STAAD.Pro Editor window.</td>
</tr>
<tr>
<td>Analysis Log</td>
<td>Opens the analysis and design results file (file extension <code>.an1</code>) in the STAAD.Pro Editor.</td>
</tr>
</tbody>
</table>
### Table 329: Tools group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Error Log</td>
<td>If analysis errors occurred, this tool is active. Click to open the error log in the STAAD.Pro Editor.</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Load Connection</td>
<td></td>
</tr>
<tr>
<td>Tag File</td>
<td></td>
</tr>
<tr>
<td>Edit Connection</td>
<td></td>
</tr>
<tr>
<td>Tag File</td>
<td></td>
</tr>
<tr>
<td>View Connection</td>
<td>Opens the Assign Connection Tags dialog.</td>
</tr>
<tr>
<td>Tags</td>
<td></td>
</tr>
<tr>
<td>Check Connection</td>
<td>(Active only after a successful analysis has been performed)</td>
</tr>
<tr>
<td>Tags</td>
<td>Opens the Assign Connection Tags dialog and Check Connection Tags dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.</td>
</tr>
<tr>
<td>Calculator</td>
<td>Opens the STAAD.Pro Calculator window, which is capable of performing mathematical operations.</td>
</tr>
<tr>
<td>Unit Converter</td>
<td>Opens the STAAD.Pro Converter window, which is used to convert data from one unit system to another.</td>
</tr>
<tr>
<td>Take Picture</td>
<td>Used to take a snapshot image of current view. The picture is automatically added to a picture album.</td>
</tr>
<tr>
<td></td>
<td><strong>Tip:</strong> The picture is stored in a STAAD-native format and can be subsequently included in custom reports during Report Setup. The Copy Picture and Export View tools may be used to copy a model image or to save it to a common file format.</td>
</tr>
<tr>
<td>Tool Name</td>
<td>Description</td>
</tr>
<tr>
<td>---------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Copy Picture</td>
<td>Used to copy the current picture to the clipboard for pasting in other Windows applications such as image editors, spreadsheet applications, or word processors.</td>
</tr>
<tr>
<td>AVI File</td>
<td>Opens the Create AVI File dialog, which is used to create a video file recording of animated deflection, section displacement, mode shape, and plate stress contour diagrams.</td>
</tr>
</tbody>
</table>

Table 330: Developer group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Macro Editor</td>
<td>Opens the STAAD.Pro Script Editor window, which is used to write macros.</td>
</tr>
<tr>
<td>Macros</td>
<td>Opens the Macro dialog, which is used to create, manage, and run Visual Basic Script macros in STAAD.Pro.</td>
</tr>
</tbody>
</table>

Table 331: User Tools group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configure</td>
<td>Opens the Customize User Defined Tools dialog (on page 3106), which is used for running a macro utilizing OpenSTAAD functions. For additional information, refer to the OpenSTAAD help section (on page 5045).</td>
</tr>
<tr>
<td>User Tools &gt; &lt;list&gt;</td>
<td>Used for running an user-defined macros. A list of all available macros is displayed in the list. <strong>Note:</strong> See To export STAAD model data in AutoPIPE (on page 882) for information on using the included Export Model to AutoPipe macro.</td>
</tr>
</tbody>
</table>
**Improper Connectivity** dialog

Used to the model for plates that are improperly connected to beams. Beams must be connected to plates at their nodes in order to ensure proper coupling and load transfer. This tool will inform the user if any improper connections are present in the model.

If there are improper connections between beams and plates (not connected by their nodes), a dialog box displaying a list of the plates and the beams intersecting them (but not connected at the node) will be shown.

Opens when the **Structure Tools > Check Beam Plate Connectivity** tool is selected in the **Geometry Tools** group on the **Utilities** ribbon tab.

- **Overlapped Entities list**
  - A list overlapping entities is displayed. Select one to highlight in the active view window by the selected option.

- **Highlight options**
  - The first option (Highlight-Both) will highlight both of selected entities listed while the last two options will highlight either beam.

- **Close**
  - Closes the dialog.

---

**Example**

![Diagram of improper beam to plate connectivity](image)

*Figure 336: An example of improper beam to plate connectivity*

---

**List of Duplicate Beams / Plates dialog**

Used to detect the presence of two or more instances of the same beam or plate, depending on the ribbon tool selected.

Opens when either **Beam Tools > Duplicate Beams** or **Plate Tools > Duplicate Plates** is selected from the Geometry Tools group on the **Utilities ribbon** tab.
If any duplicates of the selected object type are detected (that is, beams that share common end nodes or plates that share all common vertices), then a list of the duplicate beam or plate pairs is displayed.

When a pair is selected, click Merge. You are then prompted to select which member or plate number to retain.

**Overlapping Plates** dialog

Used to determine if two or more plates have intersecting boundaries not attached at nodes or otherwise overlap in the same plane.

An overlapping plate is defined as a plate whose nodes intersect other plates at points other than the defined nodes. This entails plates whose boundaries with adjacent plates are not attached at the nodes or plates within other plates (in the same plane). However, if two elements are such that the plane of one element is at an inclination to the plane of the other, and none of the nodes of one element lie on the plane of the other, those cannot be detected.

![Figure 337: Examples of overlapping plates](image)

**Tip:** Typically, a more refined meshing of the overlapping elements is required to eliminate the above problem.

**Overlapped Entities** list

A list overlapping plate pairs is displayed. Select one to highlight in the active view window by the selected option.

**Highlight options**

The first option (Highlight-Both) will highlight both of the plates listed while the last two options will highlight each individual plate separately.

**Display/Remove Dimensions** dialog

Used to display dimensions of beams in the view.

Opens when the **Beam Tools > Dimension Beams** tool is selected in the **Geometry Tools** group on the **Utilities** ribbon tab.

Dimensions can be displayed or removed from the entire structure, selected beams, or a list of beam numbers.

**Related Links**
- **M. To display beam lengths** (on page 893)

**Section** dialog

Used to cut a section through the structure along a specified global plane at a desired location of the third axis. Sectional views parallel to global XY, YZ, and XZ planes may be created.
Opens when the **Structure Tools > Cut Section** tool is selected from the **Geometry Tools** group on the **Utilities** ribbon tab.

**Range By Joint tab**
Select the plane of the section by selecting either the **X-Y Plane**, **Y-Z Plane**, or **X-Z Plane** option. In the **With Node #** drop down list, select the joint which lies on the sectional plane.

For example, if the section is to be drawn parallel to XY plane at Z=5, then, select the **X-Y Plane** option and select a joint number with a Z coordinate of 5.

**Range By Min/Max tab**
Select the plane of the section by selecting either the **X-Y Plane**, **Y-Z Plane**, or **X-Z Plane** option. The **Minimum** and **Maximum** fields represent the boundary distances along the axis perpendicular to the sectional plane. Every object lying between these two distances would be displayed.

For example, if the section is to be drawn parallel to XY plane at Z=5, select the **X-Y Plane** option and select a **Minimum** value of 4 and a **Maximum** value of 6.

**Select to View tab**
Used to specify the portion of the structure to view by using selected objects.

- **Window/Rubber Band** allows you to select the portion of the structure to view by specifying a rubber-band window around it.
- **View Highlighted Only** displays only the selected (highlighted) objects on screen. You must select the members and elements to view before choosing this option.
- **Select To View** allows you to view only Beams, Plates, and/or Solids, depending on the selection of the corresponding options.
Create Group dialog

Used to cluster a set of joints, beams, plates or solids into a single entity identified by a distinct name.

Opens when the Groups tool is selected in the Geometry Tools group on the Utilities ribbon tab.

The group name may be subsequently used instead of a joint, member, or element list to specify properties, supports, loads, etc. This prevents the need for you to repeat the same list of objects within your model. Additionally, group names can be used to highlight geometry using the Group Selection tool.

For example, the top chord members of an truss can be grouped into a name called “_TOPCHORD”.

Create

Opens the Define Group Name dialog, which is used to assign a group name and specify Node, Beam, Plate, Geometry, or Floor from the drop down list, as shown below.
Caution: Group names must begin with the underscore \_\_ character.

Delete
This button allows us to delete any group that has already been created. The names of all available groups are displayed along with a reference number in a list box. Select the one we wish to remove and click on the Delete button. That group will be removed from the list.

Highlight
This button allows us to highlight the structural components which comprise a given group. From the list box where the available group names are displayed with a reference number, select the group we wish to highlight.

Close
Click group specification is complete.

Assign Methods
Options:
- **Associate to View** – assigns all geometry shown in the current view to the group
- **Associate to Selected Geometry** – assigns selected (highlighted) geometry to the group
- **Associate to List** – assigns a typed list of joint, member or element numbers to the group

Associate
Click to add the joints, members, or elements to the group.

Note: Only the objects in the current view, selection set, or list will be added to the selected Group name. Any objects previously in that group that are not in the association method will no longer be contained in that group.

Related Links
- *M. To create a group from a selection* (on page 704)

Check Connection Tags dialog

Used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

Opens when **Connection Tags > Check Connection Tags** is selected from the right-click pop-up menu when a member is selected.
Select Load Case Types
Select All load cases, only Primary Load Cases, or only Load Combinations to display in the list.

Select Load Cases
Select one or more load cases from the list by setting the individual check boxes. You can set the Select All check box to select all displayed load cases (those filtered by the Select Load Case Types selection). To clear all load cases selections, set the Unselect All check box.

Check
Checks analysis results at each member end with a connection tag assigned from the selected set of beam members against the connection capacities defined in the Connection Tag XML file. A table with the results of the check opens.

Close
Closes the dialog.

Related Links
- D. Connection Tags Capacity Checks (on page 985)
- D. To check connection tags (on page 985)

**STAAD.Pro Calculator utility**

STAAD.Pro includes an on-screen calculator which is displayed in a separate window.

Opens when the Calculator tool is selected from the Tools group on the Utilities ribbon tab.

The calculator functions similar to a handheld calculator.

Normal, Engineering, and Integer mode are available.

**Create AVI File dialog**

Used to create a video file recording of animated deflection, section displacement, mode shape, and plate stress contour diagrams.

Opens when the Create AVI File tool is selected in the Tools group on the Utilities ribbon tab.
Total No. of Frames

Sets the number of frames used for the entire animation in the AVI file. This value divided by the Frame Rate /sec is the total length of the animation, in seconds.

Frame Rate /sec

The number of frames displayed per second of animation. In other words, this is the speed at which the animation is displayed.

Animation Type

Select the analysis results to be animated from the available options:

- **Deflection** — animates the deflected shape of the currently selected load case or combination.
- **Sectional Displacement** — animates the local displacement of members.
- **Mode Shape** — animates the deflection of the selected mode. Only active if the modes have been requested (on page 3030) in the current STAAD input file.
- **Stress Contour** — animates the selected Plate Stress Type or Solid Stress Type. Only active if Plates or Solid elements are included in the current STAAD input file.

OK

Opens a Save As dialog, which is used to specify a location and file name for the AVI file. Click OK and Cancel

Closes the dialog without creating an AVI file.

**Video Compression dialog**

Used to select the video compression codec and set associated compression parameters for an AVI file. AVI files can be quite large, and compression is a technique by which the size of these files may be reduced.

Opens when OK is clicked in the Create AVI File dialog (on page 3101).

**Compressor**

Select one of the included means of video compression:

<table>
<thead>
<tr>
<th>Compressor</th>
<th>Description</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full Frames (Uncompressed)</td>
<td>No video compression is used. This results in the largest file size but does not require any additional settings or video codecs installed.</td>
<td>none</td>
</tr>
<tr>
<td>Microsoft RLE</td>
<td>The Microsoft Run Length Encoding codec is supported in Windows Media Player 9 and higher as well Microsoft Windows XP SP2 and newer.</td>
<td>Compression Quality</td>
</tr>
<tr>
<td>Microsoft Video 1</td>
<td>Uses the Microsoft Video 1 Compressor.</td>
<td>Compression Quality</td>
</tr>
<tr>
<td>Compressor</td>
<td>Description</td>
<td>Settings</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Intel IYUV codec</td>
<td>An uncompressed video codec. The compression quality is set to 75%.</td>
<td>none</td>
</tr>
<tr>
<td>Cinepak Codec by Radius</td>
<td>The Cinepak video codec is supported in Windows Media Player 9 and higher as well Microsoft Windows XP SP2 and newer.</td>
<td>Video can be compressed to color or black &amp; white. Compression Quality</td>
</tr>
<tr>
<td>VMnc v2</td>
<td>The VMware lossless (uncompressed) codec</td>
<td>none</td>
</tr>
<tr>
<td>TechSmith Screen Capture Codec</td>
<td>Uses the TechSmith Screen Capture Codec, which provides high compression rates with little video degradation for screen recordings.</td>
<td>Compression Control</td>
</tr>
</tbody>
</table>

**Compression Quality**

Use the slider control to set the compression percentage. The lower the number, the higher the compression amount but with reduced video quality. The default setting for each Compressor is recommended. A value of 100 means no compression is used.

**OK**

Accepts the settings and produces the AVI video. A message dialog displays the status of the video creation and the animation opens in the AVI Player window.

**Cancel**

Closes the dialog without creating the AVI video.

**Configure**

(For select codecs only) Opens a dialog with additional video compression settings, as described in the table above.

**About**

(For select codecs only) Opens a dialog displaying additional license and contact information about the codec supplier.

**Macro dialog**

Used to manage VB macros in STAAD.Pro.

Opens when the Run VB Macros tool is selected.
The first box displays the name of the currently selected macro file, if one is selected. The list box displays a list of all linked macro files.

Displays a brief description of the macro, if available.

Opens the Select New Macro File Name dialog, which is used to specify a file name, file type, and description for an empty VB macro file. The Macro editor window then opens, which used to record or edit macro commands.

Active only when no macro is selected.

Opens the Macro editor window, which is used to record or edit macro commands.

Opens the Add an existing Macro dialog, which is used to select macro for linking to STAAD.Pro.

Removes the macro from the linked macros list.

The macro file is not deleted from the disk, only the link association in STAAD.Pro.

Runs the selected macro.

Closes the dialog.

**Select New Macro File Name dialog**

Used to create a new VB macro file for use in STAAD.Pro.

Opens when either Edit > Create New VB Macro... is selected or Create New is clicked in the Macro dialog (on page 3103).
Save in
Select the drive and folder where the new macro file will be saved. Several common Windows folder navigation tools are provided.

File list
A list of existing macro files is displayed here.

File name
Specify the file name in this text box.

Save as type
VB macros can be saved in one of two file types, depending on the level of file protection you want.
- A VBS macro file is a standard macro file, the contents (code) of which can be viewed by other users in any standard text editor.
- A VBZ macro file is a protected macro file, the contents of which cannot be viewed even in an external editor like Notepad. This is useful when you want to sell the macro or simply protect its contents from unintentional editing as part of a quality control program.

Description
Used to add a brief description of the macro's function. This is helpful in identifying similarly named macros.

Create New
Creates the file with the specified information and opens the file in the STAAD.Pro Macro Editor window.

Cancel
Closes the dialog without creating a new macro file.
Customize User Defined Tools dialog

Used to customize the tools which appear in the User Tools drop-down menu.

Opens when **Tools > Configure User Tools** is selected.

![Customize User Defined Tools dialog](image)

<table>
<thead>
<tr>
<th>Tool</th>
<th>What it Does</th>
</tr>
</thead>
<tbody>
<tr>
<td>New tool</td>
<td>Adds an empty entry to the Menu Items list. Type a name for a new menu item.</td>
</tr>
<tr>
<td>Delete</td>
<td>Removes the current tool from the User Tools list</td>
</tr>
<tr>
<td>Move Up</td>
<td>Moves the selected tool up in the Menu Items list</td>
</tr>
<tr>
<td>Move Down</td>
<td>Moves the selected tool down in the Menu Items list</td>
</tr>
</tbody>
</table>

**Command**

Type the path and filename of the linked macro for the menu item. Click [...] to open a file Select File dialog, which is used to locate and select files.

**Arguments**

Specify any additional parameters associated with the macro, if required.

**Initial Directory**
Ribbon Control Reference
Utilities tab

**OK**  Saves the changes and closes the dialog.

**Cancel**  Closes the dialog without saving any changes since the Apply was clicked.

**Apply**  Saves any changes made in the dialog.

**Help**  Opens the STAAD.Pro Help contents.

---

**Export STAAD Model to SACS**

Used to export the current STAAD.Pro model to a SACS model, which can be opened in the SACS system Interactive Modeling program.

Opens when the **User Tools > STAAD to SACS Export** tool is selected in the **User Tools** group on the **Utilities** ribbon tab.

- **Staad Input File**: Displays the path and filename for the current STAAD input file.
- **SACS Model File**: The file path and file name for the SACS model file (file extension .inp) to which you want to export STAAD.Pro data.
- **[...]**: Opens a Windows **Save As...** dialog box, which is used to navigate your computer’s storage and provide a file name.
- **Title**: Specify a model title for use in the SACS file.
- **About**: Displays version information about the macro behind this export facility.
- **OK**: Closes the dialog and generates the SACS file.
- **Cancel**: Closes the dialog without generating a SACS file.

---

**Export STAAD Model to AutoPIPE**

This feature is not yet documented.
Related Links

- M. To export STAAD.Pro model data into AutoPIPE (on page 882)

# Piping tab

## Table 332: Models group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Import</td>
<td>Opens a <strong>Open</strong> dialog with the <strong>Files of Type</strong> filter set to PipeLink files (file extension .pipelink), which are piping data files created by the PipeLink utility to transfer data between Bentley AutoPIPE (V8i SELECTseries 2 and greater) and STAAD.Pro (V8i SELECTseries 2 and greater).</td>
</tr>
<tr>
<td>Tool Name</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Export</td>
<td>Opens the Export Revised Model dialog (on page 3110), which is used to specify the user name and optional comment associated with the updated data set being sent from STAAD.Pro.</td>
</tr>
<tr>
<td>Reload</td>
<td>Reloads the current PipeLink database and initiates the Support Connection Wizard. Any changes made to the active model since the last save action will be discarded. Only the current active model is reloaded. Note: A reload is always performed when a model is re-imported.</td>
</tr>
<tr>
<td>Remove</td>
<td>Used to remove all imported pipe data and transferred loads from the STAAD.Pro project. A warning message will appear to confirm you wish to proceed with removing the pipe model data. Caution: This action cannot be undone.</td>
</tr>
<tr>
<td>Save</td>
<td>Saves the current state of the active pipe model. Note: Any changes to the currently active model are saved to the local database whenever the pipe model is left, the model is deactivated due to another model becoming active, an export operation is initiated, or the Save tool is selected on the Piping ribbon tab.</td>
</tr>
<tr>
<td>Pipe Models</td>
<td>Opens the Pipe Model dialog, which is used to select the active pipe model for the Piping workflow. Job and import information for the active model is also displayed here.</td>
</tr>
</tbody>
</table>

### Table 333: Connection group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="..." alt="Icon" /></td>
<td>Opens the Support Connection Wizard (on page 3113), which is used to establish logical data links between pipe stress model data imported from a PipeLink file and the current STAAD.Pro model.</td>
</tr>
</tbody>
</table>
## Ribbon Control Reference

### Piping tab

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Connect Support" /></td>
<td>Used to specify a connection between a piping model support and the STAAD.Pro structural model. Once this item is selected, the mouse pointer changes to the support connection cursor ( ), which is used to draw a support association between piping model supports and structural model nodes.</td>
</tr>
<tr>
<td><img src="image" alt="Pipe Supports" /></td>
<td>Opens the Pipe Supports table, which identifies supported pipe nodes and details each support.</td>
</tr>
</tbody>
</table>

### Table 334: Loading group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Transfer Loading" /></td>
<td>Opens the Transfer Pipe Reactions to Structure Model dialog, which is used to specify STAAD.Pro load data for importing pipe support load data.</td>
</tr>
</tbody>
</table>

### Export Revised Model dialog

A database may be created for importing into AutoPIPE. This database includes the pipe information, the structural model (including loads and analysis results), and the connection and load mapping information. This database is not constructed by the STAAD.Pro but rather the PipeLink application.

**Note:** The STAAD model should be analyzed (if loaded at all) prior to the export. Otherwise, a warning dialog will appear instructing you to do so.

Opens when **Piping > Export** is selected.

---

*STAAD.Pro 3110 User Manual*
User Name  Typically, enter the name of the engineer responsible for the structural analysis. The current Windows user name is specified here by default.

Comments  (Optional) Enter any comments regarding actions performed for reference purposes.

OK  Opens the PipeLink for STAAD.Pro V8i application, which is used to select the AutoPIPE data file for exchanging database information.

Note: Refer to the PipeLink manual for assistance in using this utility application.

Related Links
- M. To export model data for use in AutoPIPE (on page 881)

Pipe Model dialog

Used to select the active pipe model for the Piping workflow. Job and import information for the active model is also displayed here.

Opens when the Piping page is selected in the Piping workflow.
**Active Model**

Used to select the current active model. Any actions taken in the Piping mode are done only on the active model.

*Note:* The view window displays all models, with the inactive models displayed in the color set in the [Diagrams dialog Pipe tab](#) (on page 3121). Supports and labels are not displayed for inactive models for clarity.

**Import Data, Description, and Revision**

The most recent revision is detailed and the date/time when the model was imported into the local database is also displayed. The revising application version number is shown along with the application name.

**Import...**

Opens a [Open](#) dialog with the [Files of Type](#) filter set to PipeLink files (file extension `.pipeLink`), which are piping data files created by the PipeLink utility to transfer data between Bentley AutoPIPE (V8i SELECTseries 2 and greater) and STAAD.Pro (V8i SELECTseries 2 and greater).

**Related Links**

-  [M. To import a piping model](#) (on page 878)
Support Connection Wizard

Used to establish logical data links between pipe stress model data imported from a PipeLink file and the current STAAD.Pro model.

Opens when:

- A PipeLink database is successfully imported or reloaded, or
- the Support Connection Wizard tool is selected in the Connection group on the Piping ribbon tab.

AutoPIPE Imported Supports page

Used to select pipe supports which were defined in the AutoPIPE model.

<table>
<thead>
<tr>
<th>Available Supports list</th>
<th>Selected Supports list</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lists all available supports which were exported from the AutoPIPE model. This list can be filtered using the drop-down list. The filtering options are All, Connected, Unconnected, V-Stop, or Anchor (relating to whether a connection to the STAAD.Pro model has been defined). When support type information is available the filter will be expanded to include these as well.</td>
<td>All supports added to this list will be imported from the PipeLink data file into the STAAD.Pro Piping mode.</td>
</tr>
</tbody>
</table>

Structure Beams page

Used to select which structure elements will be used.
Available Beams list  The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group, View, and Property.

Selected Beams list  All members added to this list will be used in the STAAD.Pro Piping mode.

Structure Nodes page

Used to select which structure nodes will be used.
Available Nodes list  The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group and View.

Selected Nodes list

Structure/Pipe Support Connectivity Parameters page
Used to set parameters for the pipe wizard to establish connections. These parameters provide a means of controlling the results returned by the wizard in attempting to find suitable support points on the structure. The default values of distance-based parameters will depend on the base unit of STAAD.Pro.
Maximum Distance between pipe support and structural supporting point

This is the sensitivity of the connection wizard when attempting to locate appropriate points on the structure to which to connect piping support nodes. The default value is 2m / 6 feet. Any potential connections beyond Max. Range will be discarded by the wizard.

Insert additional/new nodes into beams where connection point lies on a beam

This parameter effects the connection of the structures rather the point finding algorithm. If set true a new node will be created at each intermediate beam point and the connection made to that rather than to the beam itself. If the algorithm finds new node points within “End Tolerance” of each other then only one new node will be added. This option is not selected by default.

Allow Non-Perpendicular Connection at End Nodes

Select this option to include the beam end nodes in the node search. Otherwise, the end nodes of a given beam will only be considered for connections perpendicular to the beam, unless they have been explicitly added to the node subset. This is only relevant if the node subset does not explicitly include the nodes at the end of members in the beam subset.

End Tolerance

To allow for differences in precision and to avoid very short beam breaks this parameter will determine at what distance from the node the perpendicular will be considered to be at the node itself. The default value is 5 cm / 2 inches.

Results page

Used to review the results of the pipe data import. If changes are necessary, you may go back to previous steps in the wizard to make changes. The connection finding routine runs in two parts. First looking at beams (length of perpendicular from beam) and then looking at nodes (straight line length). The five closest found connectable points are saved to be presented on the results page.

The results will be presented in a table with columns for pipe node id and for the structural item it is to be connected to. This second column will provide a drop list allowing you to choose either No connection or one of
up to 5 closest items. The items will be listed with the closest at the top. The initial state will be the closest item or Ground, if no good matches were found.

**Note:** Pipe-structure links are not part of the undo system. Nodes created at the end of the wizard will be removed by an undo but the links are not changed.

<table>
<thead>
<tr>
<th>AutoPIPE Support</th>
<th>Staad supporting point</th>
</tr>
</thead>
<tbody>
<tr>
<td>A00 : Anchor</td>
<td>Beam 24</td>
</tr>
<tr>
<td>A01 : V-Stop</td>
<td>Ground</td>
</tr>
<tr>
<td>A09 : Anchor</td>
<td>Ground</td>
</tr>
</tbody>
</table>

< Back  Return to the previous import wizard page.

Next >  Advance to the next page in the import wizard.

Finish  (Results Page only) Accepts the import settings and closes the wizard. The pipe data is displayed in the active view window.

Cancel  Closes the wizard without importing any pipe data.

Help  Opens the STAAD.Pro help window.

**Related Links**
- *M. To import a piping model* (on page 878)
- *M. To use the Support Connection Wizard* (on page 879)

**Pipe Supports table**

Identifies supported pipe nodes and details each support.

**Note:** The rows are initially sorted by Pipe Node. Clicking in the headers for Support or Type will sort on that field. As the Pipe Node button is used to select the entire contents it is not possible to return to Pipe Node sorting without using the pop up menu.

Opens when the **Supports** page is selected in the **Piping** workflow.
Tip: Multiple Pipe Nodes (rows) can be selected by holding down the <CTRL> key while making selections.

### Pipe Node / Support / Type

Each imported Pipe Node is listed in the table, along with the associated Support ID and support Type. These fields are imported from the pipe stress model and may not be edited.

### Connected to

The type of model entity to which the Pipe Node is connected. If set to Ground (default when the wizard cannot establish a connection), then the Pipe Node is assumed as not supported by the structure. You may manually connect the Pipe Node here by selecting either Beam or Node.

**Note:** This will be updated automatically if the Connect Supports tool is used.

### Structure Entity

The Beam or Node number (as specified in the Connected to column) to which this Pipe Node is connected.

### Dist. Along Beam

(Connected to Beam only) The distance from the start node of the beam where the pipe node is connected, in the current length units.

### Lever Arm

Select this option to indicate that the Pipe Node connection is a Lever Arm and thus capable of transferring the pipe loads not only as force but also as a moment. If this option is not selected, the loads transferred to STAAD.Pro will not consider the connection rigid for moment transfer (default). This is analogous to setting the connection as “fixed” versus “pinned” in STAAD.Pro for the generated forces applied to the structural model.

For example, this option would be selected for the case of an intermediate pipe anchor where the connection is rigidly attached at both pipe and structure ends or the for case when the connection at the structure end has a moment resisting design.

### Right-click Pop-up Menu

Contains tools used for controlling connections and sorting the Pipe Supports table.

<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
<th>Save Effect As…</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set All as Lever</td>
<td>Sets the option for all Pipe Nodes as Lever Arms in the active model.</td>
<td></td>
</tr>
<tr>
<td>Set All as Not Lever</td>
<td>Removes the Lever Arm option for all Pipe Nodes in the active model.</td>
<td></td>
</tr>
<tr>
<td>Set Selection as Lever</td>
<td>Sets the option for the selected Pipe Node(s) as Lever Arms.</td>
<td></td>
</tr>
</tbody>
</table>
Menu Item | Description | Save Effect As…
--- | --- | ---
Set Selection as Not Lever | Removes the Lever Arm option for the selected Pipe Node(s). |  
Disconnect All | Removes the connection for all Pipe Nodes in the active model |  
Disconnect Selection | Removes the connection for the currently select Pipe Node(s). |  
Sort by Pipe Node | Sorts the table by the Pipe Node name (default) |  
Sort by Pipe Support | Sorts the table by the name in the Support column. | Double-clicking the Support column heading  
Sort by Support Type | Sorts the table by the name in the Type column. | Double-clicking the Type column heading  

Related Links
- *M. To draw connections between piping supports and the structure* (on page 880)
- *M. To manually specify connections between piping supports and the structure* (on page 880)

**Transfer Pipe Reactions to Structure Model** dialog

Used to specify STAAD.Pro load data for importing pipe support load data.

Opens when *Piping > Transfer Loadings* is selected.

The table contains the following columns:

- **Pipe Loading** - Each pipe support is included as a separate row, with the nature of the load and load number described here.
- **Action** - Set to Create (for generating a new load) or No Action (which does not result in a new load).
  
  If the loads are being updated, additional options to Update or Remove existing loads are provided.
- **Class** - Specify if the load is a Primary load case or Reference load case type.
Tip: You can quickly set all new loads (Action = Create) to either Primary or Reference class by right-clicking anywhere along the column heading and selecting the appropriate option from the pop-up menu.

- **STAAD Load** - The load case number which will be assigned to the load.
- **Type** - You can classify the load type, which is used for automatic load combination generation.
- **STAAD Load Name** - A description load name is generated here. You may enter a different title if needed.

**OK**

Closes the dialog and proceeds with the load data transfer. A message dialog opens to provide you with a summary of the transfer process.

**Note:** Warnings are displayed in orange and errors are displayed in red.

**Cancel**

Closes the dialog without transferring any load data.

**Related Links**

- *M. To transfer load data for structural analysis* (on page 881)

**Merging Support Connection dialog**

Used to determine which of the conflicting connection definitions to use. All data contained within the table is read-only except for the **Keep Local** options for each pipe node.

Opens when clashes are detected during the import or re-import process.

**Pipe Supports data table**

Select the **Keep Local** option for any pipe node support which you do not wish to be overwritten during the import process.

**Cancel**

Closes the dialog and cancels the import process.
**OK**

Accepts the clash decisions made in the **Keep Local** choices selected and proceeds with the data import.

**Related Links**

- *M. To import a piping model* (on page 878)

**Diagrams** dialog **Pipe** tab

An additional set of view controls is available to you in the Piping mode. This tab is only active when STAAD.Pro is in the Piping mode.

**Note:** The Structure tab, Loads and Results tab, Scales tab, and Labels tab from the Modeling mode are available in the Piping mode.

Opens when **View > Structural Diagrams** is selected (Piping mode only).

![Diagrams dialog](image)

**Tip:** 3D Rendering of Piping models can be viewed using the 3D Rendering features of STAAD.Pro by either toggling on the Full Sections view in the Diagrams dialog or by using the 3D Rendering view.

**Labels**

Toggles the display of pipe support and node labels.

**Colors**

Used to select colors for pipe model elements.

**Inactive Models**

Controls the color for inactive models.

No support or node data is displayed for inactive models.
# Bridge Deck tab

## Table 335: Deck group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="create_deck_icon.png" alt="Create Deck" /> Create Deck</td>
<td>Used to create a deck definition from the selected structure geometry. <strong>Note:</strong> When a deck is defined from beam elements, the program will generate a surface of fictitious, triangular plate elements which are used in determining the load distribution. These elements have no inherent stiffness and are not transferred to STAAD.Pro for analysis of the structure. These plate elements may be reviewed by toggling on their display in the Diagrams dialog Deck tab (on page 3137).</td>
</tr>
<tr>
<td><img src="delete_icon.png" alt="Delete" /> Delete</td>
<td>Opens the Rename Deck dialog.</td>
</tr>
<tr>
<td><img src="select_deck_icon.png" alt="Select Deck" /> Select Deck</td>
<td>Opens the Select Plates in Deck dialog (on page 3124), which is used to select the members and plates associated with a previously created deck definition.</td>
</tr>
<tr>
<td><img src="define_roadway_icon.png" alt="Define Roadway" /> Define Roadway</td>
<td>Opens the Roadways dialog (on page 3123), which is used to create and manage roadway definitions.</td>
</tr>
</tbody>
</table>
Table 336: Loading group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Influence Surface Generator</td>
<td>Initiates the STAAD Analysis and Design engine to generate influence surfaces for the structure.</td>
</tr>
<tr>
<td><strong>Note:</strong> When selected, the STAAD Analysis and Design dialog opens to display the progress of the analysis used in influence surface generation. This dialog automatically closes upon completion.</td>
<td></td>
</tr>
<tr>
<td>Influence Diagram</td>
<td>Opens the Diagrams dialog Influence tab (on page 3136), which is used to control the display of influence diagrams on the defined bridge deck.</td>
</tr>
<tr>
<td>Run Load Generator</td>
<td>Opens the Load Generator Parameters dialog (on page 3130), which is used to select design codes, specify code relevant data, and select points of interest for placing loads.</td>
</tr>
<tr>
<td>View Results file</td>
<td>Opens the results file (file extension .bva) in a text editor for your review and for use in post-processing.</td>
</tr>
<tr>
<td>Browse Load Generator Results</td>
<td></td>
</tr>
<tr>
<td>Create Loading in Staad Model</td>
<td>Used to create a primary load cases in the current STAAD input file for each action requested in the Load Generator Parameters dialog (on page 3130). These load cases can then be used in the same way any other static load case would be used for analysis and design.</td>
</tr>
</tbody>
</table>

Table 337: Vehicle group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Database</td>
<td>Opens the Vehicle Database dialog (on page 3138), which is used to display code specified vehicle load data and to create user-defined loads.</td>
</tr>
</tbody>
</table>

**Roadways dialog**

Used to create and manage roadway definitions. This should be the clear roadway width between curbs or other barriers. The program will determine the number of traffic and design lanes available in this width.

Opens when Deck > Define Roadways is selected.
**Roadways list**  A list of each roadway definition for the selected deck is contained here. Selecting a Roadway definition in the list will display the characteristic parameters in the Description block below.

**New…**  Opens the Define Roadway dialog (on page 3123), which is used to parametrically define straight, curved, or custom roadways.

**Edit…**  Opens the Define Roadway dialog (on page 3123), which is used to edit the selected roadway definition.

**Delete**  Deletes the selected Roadway definition.

**Caution:**  No confirmation is required to delete a Roadway definition. This action cannot be undone.

**Close**  Closes the Roadways dialog.

**Select Plates in Deck**  dialog

Used to select the members and plates associated with a previously created deck definition.

Opens when **Deck > Select Deck** is selected.
decks list
Lists all deck definitions in the current STAAD project.

Close
Closes the dialog.

Selection Type
Used to either Select from list by cursor, which allows you to select the deck by name from the decks list, or to Select using type list, which allows you to type a list of deck names in the Enter list.

Select Listed Entities
Used with the Select using type list option to select the entities specified in the Enter list.

Enter list
Specify the names of decks here.

**Define Roadway** dialog

Used to define roadways for vehicle loading on a selected deck. Either Straight, Curved, or Custom roadways may be defined.

Opens when the New or Edit buttons are clicked in the Roadways dialog (on page 3125).
Straight tab

Used to parametrically define a straight roadway.

**Curb A / B Origin**

Curb A is the closes to the origin, with Curb B being the other side of the roadway. Dimensions are all taken from the origin (i.e., the distance to Curb B is not taken from Curb A).

**Angle**

The angle of the roadway taken clockwise from the positive X axis.

**Spacing between points**

Spacing ... in current length units.
**Curved tab**

Used to parametrically define a curved roadway.

**Center**  
Define the center of a circular arc.

**Direction**  
Select if the arc sweep is defined as Clockwise or Anticlockwise (Counter-clockwise) from ...

**Radius**

**Angle**

**Spacing between points**  
Spacing ... in current length units.

**Custom tab**

Used to define a generic roadway section.
Add Lane Left
Copy Lane Left
Add Lane Right
Copy Lane Right

**Active Lane**
Select the lane you wish to create or edit from the drop down list. The total number of lanes present is also displayed.

**Origin X / Z**

**Width**

**Delete Lane**

**Add Section**

**Section Type**
Select if the roadway segment is **Straight**, **Curved**, or **Custom**. The section panel updates to allow entering parameters for this selection.
• Straight section
  
  Active Section
  Delete Section
  Curb on Left / Right
  Select either or both of these options to ...
  Orientation
  Length

• Curved section

  ![Curved section screenshot]

• Custom section

  ![Custom section screenshot]

**OK**  Closes the dialog and creates a new roadway or updates changes made to an existing roadway.

**Cancel**  Closes the dialog without saving any changes or creating a roadway definition.

**Apply**  Updates the current roadway definition with any changes made in the dialog.

**Help**  Opens the STAAD.Pro help window.
Load Generator Parameters dialog

Used to select design codes, specify code relevant data, and select points of interest for placing loads. Opens when Loading > Run Load Generator is selected.

![Image of Load Generator Parameters dialog]

**General tab**

**Design Code** Select the specification from which the loadings are to be taken.

*Note:* The `<code>` tab updates accordingly with this choice.

**Limit State** Specify if the vehicle loads are to be used with the Ultimate or Serviceability limit state.

**Decks tab**

Used to select the previously defined decks to be considered for loading. A check is placed in the box associated each deck name to be considered.
<code>tab

Used to input code-specific parameters. This tab dynamically updates based on the Design code selection made on the General tab.

Note: The following pages provide details on these code-specific parameters.

Node Displacements / Support Reactions / Beam End Forces/ Plate Center Stress tabs

Used to specify a structural object, action, and effect as a point of interest for influence surfaces which determine the placement of the design loads on the structure to achieve maximum (or minimum) action.
Node / Beam / Plate  Specify the number of the structural object to act as a point of interest. The point of interest is the location on the structure the resulting action at which will be used to determine the placement of the loads.

Displacement / Support Reaction / Stress Force  The action of interest. This is usually the direction of the displacement, force, moment, or stress.

- In the case of beam end forces, you must select the member end as well as the direction of force or moment.
- In the case of plate stresses, additional options for the maximum combined effect (von Mises or Principle stresses) are also available.

Effect  Select either the maximum (+ve) or minimum (-ve) effect at the point of interest due to the selected action.

OK  Accepts the specified parameters and closes the dialog.

Cancel  Closes the dialog without adding load generation parameters.

Apply  Applies the parameters from the current tab.

Help  Opens the STAAD.Pro help window.

BS 5400 Specific Parameters

Used to input parameters for loading per Cl.6.3.1 of British Standard 5400 “Steel, concrete and composite bridges, Part 2. Specification for loads”.

<table>
<thead>
<tr>
<th>Combination</th>
<th>Vehicle Id</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>BS5400HBNominal</td>
<td>30</td>
</tr>
</tbody>
</table>

**Combination**  Select the load combination number to be considered which include highway bridge live loading (BS 5400:Part 2 table 1). Refer to Cl.6.2.7 and 6.3.4 for the load factors used in the program for the load combinations.

**Vehicle table**  Select the design vehicle to use. The default is the HB vehicle per Cl.6.3.1. User defined vehicles may also be selected.

Specify the number of units of 10 kN load per axle to be used for type HB to be used per Cl.6.3.
**BD21/01 Specific Parameters**

Used to input parameters for loading per BD21/01 “The Assessment of Highway Bridges and Structures”.

**Loading Class**  
Select the Assessment Loading vehicle from this list per Cl.5.12 through 5.17 of BD21/01.

**Road Class**  
Select the bridge situation in terms of road surface characteristics and traffic level per Cl.5.22 of BD21/01.

**BD21/01 Annex D Specific Parameters**

Used to input parameters for loading per Annex D: Loading From Vehicles of BD21/01 Annex D. This code is used for traffic on cross girders and transversely spanning slabs.

*Note: If a fire engine convoy is used then there must be at least two normal vehicles. Axle Impact Factor is only used when there is one normal vehicle and no fire engines.*
Fire Engine
Select the make of fire engine from Table E1 of BD21/01 Annex E as the Type. Specify the Number in Convey to indicate the number of fire engines (all same type) with which to simultaneously load the bridge.

Nominal Vehicle
Select the nominal vehicle Type from Tables D1 and D2 of BD21/01 Annex D.

Vehicle nomenclature:
BD21/01_WT[#][dir]
Where:

WT Either the vehicle gross weight value (Table D1 vehicles) or the vehicle reference (Table D2 vehicles).

# Separate id number for vehicles which are listed more than once in the table.

div a or b signifies a vehicle in which the W2 and W3 loads have exchanged position.

Specify the Number of vehicles to indicate the number of nominal vehicles (all same type) with which to simultaneously load the bridge. Specify an Axle Impact Factor for use with a single vehicle per Cl.D.3(a).

AASHTO Specific Parameters
Used to input parameters for loading per “Standard Specification for Highway Bridges”, AASTHTO ASD / LFD.

Loading Class
Select the nominal vehicle load to use per Cl.3.7. All vehicles in the Vehicle Database dialog are listed here, with the following being provided in the AASHTO ASD/LFD code:

- H15-44
- H20-44
- HS15-44
- HS20-44
AASHTO LRFD Specific Parameters

Used to input parameters for loading per the AASHTO LRFD Bridge Design Specifications.

- **Live Load Impact Factors**: Multiplier accounting for imperfection in road surface resulting in additional dynamic loading. These factors vary dependant on the bridge component being designed. The default value of 1.33 is specified for bridge components (other than deck joints) for limit states other than fatigue or fracture (AASHTO LRFD table 3.6.2.1-1). Separate values may be specified for the **Design Tandem** vehicle and the **Design Truck (HS)**.

- **Multiple Presence Factors**: Multipliers for design vehicles based on the number of design lanes in the transverse section of a roadway. The multiple presence factors, m, account for the statistical improbability of three or more adjacent lanes being loaded simultaneously with the design vehicle. The default values are taken from AASHTO LRFD table 3.6.1.2-1.

IRC Specific Parameters

Used to input parameters for loading per IRC Chapter 6.

The IRC rules have been implemented such that the program will select the appropriate live loads, number of design lanes, and generate all the possible load combinations as stated for a specified roadway width.
Loading Class
Select the vehicles to be used for this load generation.

Impact Factor
Specify a value to account for dynamic effects resulting for road surface imperfections.

Reduction in longitudinal effect
Contains a table of Multiple Presence Factor for the number of lanes for a specified roadway.

Related Links
- *AD.2007-07.3.5 Transverse IRC Loading in STAAD.Beava* (on page 170)

Diagrams dialog

The diagrams dialog contains additional tabs in the Bridge Deck mode used to control the display of decks and influence diagrams.

Opens when **View > Structural Diagrams** is selected.

Influence tab

Used to control the display of influence diagrams on the defined bridge deck as displayed on the Deck | Deck Definition page.

Opens when:
- **View > Structure Diagrams** is selected, or
- **Loading > Influence Diagrams** is selected.
Diagram Type
Select the type of action you wish to display in a contour plot the deck. The <action type> and <structural object> fields update with the selected type.

[action type]
Select the action and direction for the action.

[structural object]
Select the node, beam, or plate for which the action will be displayed.

Influence Show
Select if the influence is drawn for All Decks defined in the Bridge Deck mode or only the Active Deck.

Tip: The Active Deck is the selected deck in the View toolbar.

Scale for Influence Draw on Beams
Adjusts the scale for influence curves drawn onto beam objects.

Deck tab
Used to control the display of bridge decks and loads on the Deck | Deck Definition page.

Opens when:
- View > Structure Diagrams is selected, or
- Loading > Influence Diagrams is selected.
**Roadways**
This option will display the defined roadways and the design lanes, as determined by the program. Different colors are available for the Active (currently selected) and Inactive elements.

**Results**
These options are used to display the Loads and Vehicles on the structure. Different colors are available for the Active (currently selected) and Inactive loads.

**Triangulation**
Used to display the triangular, pseudo-plate elements created when a deck is defined from beam members only.

**Vehicle Database** dialog

Used to display code specified vehicle load data and to create user-defined loads.

**Note:** Vehicles included in Bridge Deck may not be edited or deleted.

Opens when **Vehicle > Database** is selected.
Vehicles list

All vehicle definitions in Bridge Deck are listed here. User-defined vehicles are added at the bottom of this list. Select a user-defined vehicle to make any changes.

New ...

Opens the Add New Vehicle dialog, which is used to specify a name for the vehicle load to be added.

Delete

(User-defined loads only) Deletes the currently selected vehicle definition.

Caution: There is no confirmation of this action and this action cannot be undone.

Rename

(User-defined loads only) Opens a Rename Vehicle dialog, which is used to specify a new name for the selected vehicle definition.

Vehicle Data

The dialog contains a set of simple parameters which can be used to define virtually any vehicle load.

- Force / Length - Select the units for use when specifying Force (lbF or kN) and Length (m, mm, or ft) values.
- Front / Rear Clearance - Specify the clear distance between the first and last axle, respectively, to another vehicle or load within the same design lane.
- Width - Specify the total width (half on either side of the vehicle center line) for placement within a design lane.
- A plan diagram displays the configuration of the vehicle load. Axle 1 is the right-most solid line, with subsequent axles to the left. Variable and multiple Fixed axle Positions are shown in dashed lines.
- Is Convoy Vehicle - Select this option to signify to the program that this vehicle is to applied repeatedly along the design lane, using the Front and Rear Clearances specified.
- Number in Convoy - Specify the number of vehicles present in the convoy.
- Axles - Use the combo box to select the number of axles (fixed or variable) on the vehicle.
• Details of Axle - Select the axle (up to the number of Axles specified) for which the current load and position data is entered.
• Wheels - Use the table to enter the Force (wheel weight) and Position (distance from vehicle center line), in the selected units, for the currently selected axle.
• Bending Axle - Select this option ...
• Positions - Select if the current axle is a Fixed distance (or a number of different fixed distances) from the previous axle or if the distance is Variable. For Fixed positions, the table below allows for one or more set distances between the current and previous axle. For Variable positions, a Minimum and Maximum distance must be specified, along with an Increment value for generating a finite number of positions.

OK  Accepts changes made to the currently selected vehicle (user-defined only) and closes the dialog.
Cancel  Closes the dialog without saving any changes.

Results tab

Table 338: View Results group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load</td>
<td>Select the active load case, load combination, or load envelope from the pop-up dialog.</td>
</tr>
<tr>
<td>View Loading Diagram</td>
<td>Click to toggle the display of the current load case on the structure.</td>
</tr>
<tr>
<td>Deflection</td>
<td>Displays the deflected shape on the structure using nodal displacements.</td>
</tr>
<tr>
<td>Dipslacement</td>
<td>Displays the nodal and beam sectional displacements.</td>
</tr>
<tr>
<td><strong>Tip</strong>: This will show a more accurate representation of the deflected shape compared to the Deflection tool, but may take longer to render for large structures.</td>
<td></td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>----------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Utilization Ratio</td>
<td>Displays the design code based utilization ratio, for code checks and member selection operations, in tabular form and annotated on the structure diagram. Note: The colors representing the pass or fail status of a member can be controlled using the Color Manager dialog and the Diagrams dialog Design Results tab.</td>
</tr>
<tr>
<td>FX</td>
<td>Displays the axial force diagram on the structure.</td>
</tr>
<tr>
<td>FY</td>
<td>Displays the shear force Y diagram on the structure.</td>
</tr>
<tr>
<td>FX</td>
<td>Displays the shear force Z diagram on the structure.</td>
</tr>
<tr>
<td>MX</td>
<td>Displays the torsion diagram on the structure.</td>
</tr>
<tr>
<td>MY</td>
<td>Displays the bending moment Y diagram on the structure.</td>
</tr>
</tbody>
</table>
### Tool name

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MZ</td>
<td>Displays the bending moment Z diagram on the structure.</td>
</tr>
<tr>
<td>Beam Stress</td>
<td>Displays the beam stress diagrams on the structure.</td>
</tr>
<tr>
<td>Plate Stress</td>
<td>Displays plate stress contours on the structure.</td>
</tr>
<tr>
<td>Solid Stress</td>
<td>Displays solid stress contours on the structure.</td>
</tr>
<tr>
<td>Layouts &gt;</td>
<td>Select one of the tables of graphs of results from the drop-down gallery.</td>
</tr>
</tbody>
</table>

### Table 339: Dynamics group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode</td>
<td>Select the mode number from the drop-down list.</td>
</tr>
<tr>
<td>Mode Shape</td>
<td>Displays the mode shape diagrams for individual modes.</td>
</tr>
</tbody>
</table>
## Table 340: Animation group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Animation</td>
<td>Opens the <strong>Diagrams</strong> dialog <strong>Animation</strong> tab, which is used to control what effect is being animated on the Animation page in the <strong>Postprocessing</strong> workflow.</td>
</tr>
</tbody>
</table>

## Table 341: Configuration group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select Load Case</td>
<td>Opens the <strong>Results Setup</strong> dialog, which is used to select the load cases, mode shapes, and structural elements (beams, nodes, elements, etc.), which are included in the Postprocessing operations</td>
</tr>
<tr>
<td>Structure</td>
<td>Opens the <strong>Diagrams</strong> dialog, which is used to customize the view of the structure by setting different view-related parameters.</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Scale</td>
<td>Opens the <strong>Diagrams</strong> dialog to the <strong>Scales</strong> tab, which is used to specify the scales to which the different diagrams for viewing input and results should be plotted.</td>
</tr>
<tr>
<td>Annotate</td>
<td>Opens the <strong>Annotation</strong> dialog, which is used to annotate the display of results annotations in the View window with numerical values.</td>
</tr>
</tbody>
</table>

**Table 342: Properties group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| Update Properties | Updates the section properties assigned to members in the input file to those which are the result of a design member selection or group member command.  
If member selection is carried out for steel, timber or aluminum sections in the model, or if a GROUP MEMBER command is specified (see TR.50 Group Specification (on page 2855)), the properties of the structure at the end of these processes will no longer be the same as what we assigned initially or started out with.  
A warning dialog opens to confirm you wish to proceed before making any changes.  
**Note:** By performing this operation, the input file gets modified. Consequently, the analysis results are no longer compatible with the modified structure.  
Therefore, the **Analytical Modeling** workflow is automatically selected so the structure can be re-analyzed. Before doing so, it is advisable to go back and replace the SELECT and GROUPing commands with CHECK CODE commands. |
Table 343: Reports group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node Displacement</td>
<td>When a node selection is made, this tool opens Node Displacement dialog, which is used to generate</td>
</tr>
<tr>
<td>Support Reaction</td>
<td></td>
</tr>
<tr>
<td>Beam Property</td>
<td></td>
</tr>
<tr>
<td>Beam End Forces</td>
<td></td>
</tr>
<tr>
<td>Section Displacement</td>
<td></td>
</tr>
<tr>
<td>Section Forces</td>
<td></td>
</tr>
<tr>
<td>Beam Stresses</td>
<td></td>
</tr>
<tr>
<td>Column Transfer Force</td>
<td></td>
</tr>
<tr>
<td>Plate Stresses</td>
<td></td>
</tr>
<tr>
<td>Principal Stresses</td>
<td></td>
</tr>
<tr>
<td>Floor Vibration Report</td>
<td></td>
</tr>
</tbody>
</table>

**Annotation dialog**

Used to annotate the display of results annotations in the View window with numerical values. For example, when a bending moment diagram is displayed in the Main Window area, we may use this option to display the values of the maximum bending moment and/or the end-moment values of selected members.

Opens when the Annotate tool is selected in the Configuration group on the Results ribbon tab.
Ranges tab
Used to select the members and nodes for which we want to display the values.

- **None**: Does not annotate any Member, Element, or Node.
- **All**: Annotates all Members, Elements, and Nodes in the current view.
- **View**: Annotates all Members, Elements and Nodes in a previously saved view. Select the desired view from list of available views using the drop down list.
- **Group**: Annotates all Members, Elements and Nodes in a previously saved group of structural elements. Select the Group option and then select the desired group from the view from list of available groups using the drop down list.
- **Property**: Select this radio button to select the members by assigned Property Reference Number and description.
- **Ranges**: Specify Node, Member and/or Element numbers. You may specify individual values separated by comma, by a list, or specify a range such as 5-10.

Beam Results tab
Used to select the values, which will be displayed on the diagram.

- **Bending**: This group of check boxes determines the Bending Moment values which will be displayed on the Bending Moment Diagram. 
  - **Ends**: option will display the end-moments of the members.
  - **Maximum**: option will display the maximum moment value.
  - **Mid point**: option will display the Bending Moment value at the mid point of the member.
- **Shear**: This group of check boxes determines the Shear Force values, which will be displayed on the Member Force diagram.
  - **Ends**: option will display the values at the two end nodes of the member.
  - **Maximum**: option will display the maximum force value.
  - **Mid point**: option will display the force value at the mid point of the member.
- **Axial**: Switch this on to annotate the axial tension or compression values at the start and end of selected members.
- **Max Resultant**: Displays the maximum resultant section displacement value for each member.
- **Combined Bending and Axial Stress**: This group of check boxes determines the stress values, which will be displayed on the Stress diagram.
  - **Ends**: option will display the values at the two end nodes of the member.
  - **Maximum**: option will display the maximum value.
  - **Mid point**: option will display the value at the mid point of the member.

Node tab
Nodal point displacements can be annotated using these options. More than one may be used simultaneously.

- **Global X / Y / Z**: Display the point displacements of a node in any of the global directions.
- **Resultant**: Displays the SRSS resultant displacement. That is, the displacement as calculated by:

\[ \sqrt{(\text{Global-X}^2 + \text{Global-Y}^2 + \text{Global-Z}^2)} \]
Reactions tab
Used to select the Annotation type for the Support Reactions diagram.

Direct/Bending  Check one or more check boxes from Global X/Global rX, Global Y/Global rY, Global Z/Global rZ to select the degrees of freedom to Annotate.

Show Lines  Check this box to display line or arc with arrowhead to show the direction of the Reactions.

Scale  Use these spin boxes to specify the Scale factors for the Reaction force and Reaction moment in the diagram.

Annotate  Applies the selected annotation(s) to the View window.

Remove All  Removes all annotations from the View window.

Close  Closes the dialog box.

Related Links
- *P. To display reactions at each support* (on page 2214)
- *P. To display reactions at each support* (on page 2214)

Beam Property dialog
Used to generate a sorted table of the selected beams for inclusion in the report.
Opens when the **Report > Beam Property** tool is selected in the **Reports** group on the **Results** ribbon tab.

Sorting tab

Sort By Properties  Select the criteria by which the selected beams will be sorted in the generated table.

Absolute Values  Select this option to sort by absolute value of the selected property.

Set Sorting Order  Select the order by which the selected property is to sorted.

Report tab

Title  Enter a title for the generated table.

Save Report  Select this option to save the properties of the generated table

OK  Closes the dialog and generates a beam table for inclusion in a report.

Cancel  Closes the dialog without generating a beam report.

Help  Opens the STAAD.Pro help window.

Results Setup dialog
Used to select the load cases, mode shapes, and structural elements (beams, nodes, elements, etc.), which are included in the Postprocessing operations.
Opens when:
• the Postprocessing workflow is first selected, or
• when the Select Load Case tool is selected in the Configuration group on the Results ribbon tab.

**Loads tab**

Used to select the Load Cases to be included for Post Processing.

The Available list on the left displays all available load cases.

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected load case to the Selected list</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all load cases to the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all entries in the Selected list, load combination is placed back in the Available list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected entry from the Selected list.</td>
</tr>
</tbody>
</table>

The Selected list displays load cases that have been selected.

**Modes tab**

For dynamic analyses including response spectrum and time history analysis and/or modal extraction commands, the Modes tab becomes activated. This tab lists all available mode shape numbers.

**Range tab**

Offers selection conditions based on the attributes of Members, Nodes, Groups, Properties, etc. These options determine whether the result tables are to be displayed for all Members/Nodes/Elements, or for the selected ones.

- **All** Selects all structural elements.
- **View** Selects the structural elements stored in a previously saved view. The drop down list displays all previously saved views for selection.
- **Group** Selects the structural elements stored in a previously saved group. The drop down list displays all previously saved groups for selection.
- **Property** Selects the structural elements having a specified property tag. The drop down list displays all existing property tags.
- **Ranges** These edit boxes allow you to select Nodes and Beams by specifying a range. Specify Node, Member and/or Element numbers. You may specify individual values separated by comma, by a list, or specify a range such as 5-10.
- **Detail Tables** The Increments option specifies the number of segments into which a member would be broken up for printing section forces, displacements, etc.

**Results View Options tab**

STAAD.Pro provides you with two choices in the size to which results such as displacements and moment diagrams are plotted:

---

*STAAD.Pro* 3148 *User Manual*
Instruct STAAD to plot the result diagrams using a scale that will best fit the diagram within the window bounds. The drawback of this approach is that the variation in result values from one load case to the next will be hard to perceive if all load cases are plotted to the same maximum size dictated by the bounds of the drawing window.

Instruct STAAD to use the same basic scale for all load cases. The drawback of this approach is that while some load cases will have their results fit within the window bounds, some others may be too hard to perceive due to their small values, and others may go out of the bounds of the screen. The user is then required to modify the scale of each plot individually to better see each plot.

Choosing between these two settings is done using this facility.

**Enable Automatic Scaling** Use this to switch on the type of scaling described in (a). If we wish to use (b) instead, leave this box unchecked.

**Displacement/Deflection/Mode Shape/Beam Results** These are the check boxes for the individual result types for which we want Auto-Scaling enabled. Any box left unchecked will be subject to the rules of method (b).

### Node Displacement dialog

Used to create a Joint Displacement report for the selected node(s).

Opens when the **Report** > **Joint Displacement** tool is selected in the **Reports** group on the **Results** ribbon tab.

#### Sorting tab

**Sort By Properties** Select the criteria by which the selected beams will be sorted in the generated table.

**Absolute Values** Select this option to sort by absolute value of the selected property.

**Set Sorting Order** Select the order by which the selected property is to sorted.

#### Loading tab

Used to select the load cases to be considered in the sorted report.

**Available List** List of available load cases in the model which will not be included in the generated table.

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected load case to the Selected list</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all load cases to the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all entries in the Selected list, load combination is placed in the Discarded Load Combinations list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected entry from the Selected list.</td>
</tr>
</tbody>
</table>
Selected list

List of loads for which the nodal displacement results will be included in the generated report.

Report tab

Title

Enter a title for the generated table.

Save Report

Select this option to save the properties of the generated table

Transfer Forces for Selected Members dialog

Used to report transfer forces across beam connections on either side of columns for a selected set of members. STAAD.Pro calculates the “Transfer force” (i.e., “pass through force”), which can be used for connection design. This feature is based on a paper on the subject by Dr. William A. Thornton.

Transfer force is simply the maximum net horizontal force that gets transferred from the one side of the column to the other through the connection. So STAAD.Pro checks the forces in the members framing into each side of the column and finds out the resultant horizontal force for either side. Typically the resultant forces on the two sides would not be equal as some amount of force will be taken up by the column in shear. The greater of the two resultants is reported by STAAD.Pro as the transfer force. The option to determine transfer force automatically, will save engineers considerable time and effort as in most cases, they have to report the transfer forces in the design drawings. The same concept can be applied to floor bracing in horizontal plane.

Note: Please refer to the example file TransferForces.std, which is available in many of the regional/country example folders contained in the C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\ folder.

Opens when the Report > Column Transfer Forces tool is selected in the Reports group on the Results ribbon tab.

Loads

Displays the load cases that has been considered to calculate the transfer force. By default, all load cases are considered but one can exclude a few of them by simply clicking on the load case within the Loads box.

Left / Right Beams

List all the members on the respective sides of the column along with their incidences. The boxes Left TF and Right TF shows the resultant horizontal force from either side. Max TF reports the transfer force.

Insert to Table

Generates the Transfer Force Report table which contains all the transfer force information and can be included in reports.

Close

Closes the dialog.

Related Links

• P. To generate transfer forces report (on page 2219)

Floor Vibration Output dialog

Used to review the floor vibration analysis per AISC Design Guide No. 11 for a specific floor system and load combination. To utilize this feature, the floor system must be defined as a composite deck in the modeling mode.

Opens when the Report > Floor Vibration Report tool is selected from the Reports group on the Results ribbon tab.
The adequacy of a floor system from the standpoint of its vibration serviceability due to human activity, specifically walking excitation, can be assessed using STAAD.Pro. The procedures of Chapters 3 and 4 of the AISC Steel Design Guide Series No. 11 - *Floor Vibrations due to Human Activity* are used by the program.

The vibration calculation is done for:

a. beam or the joist mode
b. girder mode

The two modes are then combined to obtain the system frequency and other results of the combined mode using the Dunkerley relationship described in chapter 3 of the AISC Design Guide. Results for the 2 basic modes and the combined mode are provided in a tabular form.

The output for the combined mode consists of:

a. the peak acceleration for walking excitation
b. allowable acceleration (known in the code as the acceleration limit)

The design criterion as stated in the code in the third paragraph in Chapter 4 is that a floor system is satisfactory if the peak acceleration does not exceed the acceleration limit.

**Select** Select a previously defined Composite Deck.

*Note: Composite Decks are defined use the Composite Deck Layout tool.*

**Load Case** Select the load case for analysis.

**Check** Performs the floor vibration checks per AISC Design Guide 11.

**Print** Opens a Print dialog, which is used to create a hard copy of the currently displayed floor vibration report.

**Related Links**
- *AD.2005.2.1 Floor Vibration Analysis* (on page 365)
- *P. Floor Vibrations Engineering Theory* (on page 2221)
- *P. To generate a floor vibration report* (on page 2219)

**Pages in the Post-Processing Mode**

The Pushover sub-pages in the post-processing mode are described here. These pages are available only upon a successful pushover analysis.
### Ribbon Control Reference

**Results tab**

<table>
<thead>
<tr>
<th>Page</th>
<th>Sub-Page</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pushover</td>
<td>Loads</td>
<td>Opens the <strong>Load Values table</strong> (on page 3152), which displays the nodal load values and total base shear for the selected load step. The magnitude and direction of the loads in the current load step are displayed in the view window.</td>
</tr>
<tr>
<td></td>
<td>Capacity Curve</td>
<td>Opens the <strong>Capacity Curve graph</strong> (on page 3153), which displays displacement vs. force for the pushover analysis. The Capacity Curve table contains the roof or control joint displacement and base shear values recorded for each load step.</td>
</tr>
<tr>
<td></td>
<td>Node Results</td>
<td>Displays nodal displacements and rotations in the <strong>Node Displacements table</strong> (on page 3154) and the support reactions in the <strong>Support Reactions table</strong> (on page 3155) for the selected load step. The deflected shape is overlaid on the structure in the view window.</td>
</tr>
<tr>
<td></td>
<td>Beam Results</td>
<td>Opens the <strong>Beam Hinge Results table</strong> (on page 3156) which displays the status of the beam and hinge conditions along each beam's length. The <strong>Beam Force Detail table</strong> (on page 3157) displays the cross-sectional forces and moments at sections along each beam's length are displayed for the selected load step. The status of beams and hinges are displayed in the view window.</td>
</tr>
</tbody>
</table>

**Pushover Load Values table**

Displays the nodal load values and total base shear for the selected load step.
Select Load Step

Use the arrow controls or type the number of the load step. The tabulated load values, base shear, and view window load display update automatically.

Capacity Curve graph

Displays the displacement versus base shear graph for the pushover analysis. The displacement at control joint is given if a control joint was specified as a solution control method. Otherwise, the displacement is taken largest Y nodes (i.e., the "roof" level).

Opens when the Pushover | Capacity Curve page is selected.
The individual values of the curve are displayed in the **Capacity Curve** table.

**Related Links**
- *G.17.4.1.6 Frame element hinge properties* (on page 2382)
- *P. To review pushover beam results* (on page 2224)

***(Pushover) Node Displacements table***

Displays the nodal displacements and rotations for the current load step.
Select Load Step Use the arrow controls or type the number of the load step. The tabulated node displacement & rotation values, base shear, and view window displacement display update automatically.

(Pushover) Support Reactions table
Displays the nodal displacements and rotations for the current load step.

Select Load Step Use the arrow controls or type the number of the load step. The tabulated reaction values and base shear update automatically.
**Beam Hinge Results** table

Displays the cross section forces and moments at 1/12th points along the beam’s length for the current load step.

![Beam Hinge Results table](image)

**Select Load Step**

Use the arrow controls or type the number of the load step. The tabulated section forces & moments, base shear, and view window hinge display update automatically.

**Table**

The table displays the overall beam status as well as the location and status of any hinges which have formed. Up to three hinges may be listed. The hinge location (Section) and Status are displayed in physical order, from start to end node, of their location along the beam.

The table contains the following columns:

- **Status** - The material status of the beam for current load step. Linear indicates no hinges have formed in the beam. Nonlinear indicates that one or more hinges have formed. Inactive means that three hinges have formed or that one or hinges are beyond the collapse prevention (CP) limit.
- **Dir (Local)** - the direction of bending for resulting in the formation of hinges.
- **Hinge Section & Status** - The location along the length of the hinge and the acceptance criteria status. The location and status of hinges are displayed graphically on the structure in the view window.
(Pushover) **Beam Force Details table**

Displays the cross section forces and moments at 1/12th points along the beam’s length for the current load step.

![Beam Force Details table](image)

**Figure 343: Beam Force Details table**

**Select Load Step**

Use the arrow controls or type the number of the load step. The tabulated section forces & moments, base shear, and view window hinge display update automatically.

**Steel AutoDrafter tab**

**Table 344: Edit group**

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Grids" /></td>
<td>Opens the Grid Manager dialog, which is used to select which plan grids and elevation marks are displayed on the structure as well as in generated drawings.</td>
</tr>
</tbody>
</table>
### Drawing Groups

Opens the Drawing Groups tab in the Drawing Lists & Groups panel.

### Drawing Style Manager

Opens the Drawing Style Manager dialog, which is used to create drawing styles for use in Steel AutoDrafter.

### Table 345: View group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node Number</td>
<td>Controls the display of node numbers in the view area.</td>
</tr>
<tr>
<td>Member Number</td>
<td>Controls the display of member numbers in the view area.</td>
</tr>
<tr>
<td>Member Property</td>
<td>Controls the display of member material names in the view area.</td>
</tr>
<tr>
<td>Grids</td>
<td>Controls the display of grids in the view area.</td>
</tr>
</tbody>
</table>
Table 346: Drawings group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>View Drawings List</td>
<td>Opens the Drawing List &amp; Groups panel in the view area. This panel is used to generate drawings individual, all drawings in the list, and to manage groups.</td>
</tr>
</tbody>
</table>

Table 347: Material Take Off group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drawing</td>
<td>Opens the material take-off for the entire structure in a drawing.</td>
</tr>
<tr>
<td>Text</td>
<td>Opens the material take-off for the entire structure in an HTML text table.</td>
</tr>
</tbody>
</table>

Related Links

- P. To open the Steel AutoDrafter workflow (on page 2227)

**Grid Manager** dialog

Used to select which plan grids and elevation marks are displayed on the structure as well as in generated drawings.

Opens when the **Grids** tool is selected from the **Edit** group on the **Steel AutoDrafter** ribbon tab.

The program will generate a grid line for nodal X and Z coordinate in the model. An elevation mark will be generated for each nodal Y coordinate.

Check the option for each grid or elevation mark to display it in the view area and in generated drawings. Change grid or elevation mark labels by typing in the associated field for each.
Ribbon Control Reference
Steel AutoDrafter tab

Dialog controls

- **Import**
  Used to import grids from an existing adfx file.

- **OK**
  Updates the grid display and closes the dialog.

- **Cancel**
  Closes the dialog without saving any changes.

Related Links

- *P. To edit grid labels* (on page 2231)

Drawing Style Manager dialog

Used to create drawing styles for Steel AutoDrafter. This facilitates production of drawings as per selected style. You can also create standard styles with the required variations.

Opens when the Drawing Style Manager tool is selected from the Edit group on the Steel AutoDrafter ribbon tab.

All aspects of the drawing; Layer Descriptions, Layer colors, Connection / Member Descriptions, Dimension styles, Text Styles and sizes, and grid styles can be changed to create new text styles.

The Drawing Style can be imported, exported, and restored from the default file.

Drawing Style can be imported from another adfx file using the import grid option provided. The adfx file will be available along with the relevant STAAD.Pro file in the same folder.

Layers tab

This table lists all the layers that are used in the drawings generated by Steel AD. The layer descriptions and the colors can be changed.

Member Labels tab

Member / Connection type table lists all the various symbols that are used in the drawings. These symbols can be changed as desired.

In case you do not wish to have any symbols in the drawing, keep the symbols column blank.

The Section Referencing box, determines how the various sections will be labeled in the drawing, by using a reference number or the entire description of the section.

Appearance tab

This tab lists all the appearance related data. Fonts, Text Heights, Dimensions Styles and Grid Styles can be set in this tab.

Related Links
Member Labels

The Default labels to be presented are as mentioned below. You can change them as required using the settings provided in the **Drawing Style Manager** dialog.

### Member Labels

<table>
<thead>
<tr>
<th>Member Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GM</td>
<td>Beam - Column Moment Connection</td>
</tr>
<tr>
<td>GS</td>
<td>Beam - Column Shear Connection</td>
</tr>
<tr>
<td>BS</td>
<td>Beam to beam Shear Connection</td>
</tr>
<tr>
<td>C</td>
<td>Column</td>
</tr>
<tr>
<td>V</td>
<td>Bracing in Elevation</td>
</tr>
<tr>
<td>HV</td>
<td>Bracing in Plan</td>
</tr>
<tr>
<td>PST</td>
<td>Post</td>
</tr>
<tr>
<td>PTL</td>
<td>Portal</td>
</tr>
<tr>
<td>TC</td>
<td>Top Chord of Truss</td>
</tr>
<tr>
<td>BC</td>
<td>Bottom Chord of Truss</td>
</tr>
<tr>
<td>PUR</td>
<td>Purlin</td>
</tr>
</tbody>
</table>

### Member Labeling System

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Memm Type</td>
<td>Section Ref No</td>
<td>Built Up Reference</td>
<td>Labels</td>
</tr>
<tr>
<td>GS</td>
<td>14</td>
<td>F - Channel F/F</td>
<td>GS14-F1</td>
</tr>
<tr>
<td>GM</td>
<td>14</td>
<td>B -Channel B/B</td>
<td>GS14-B2</td>
</tr>
<tr>
<td>GM</td>
<td>10</td>
<td>TB- 2 Cover Plates on I/H sxn</td>
<td>GM10-TB1</td>
</tr>
<tr>
<td>GM</td>
<td>10</td>
<td>TC- 1 Cover Plates on I/H sxn</td>
<td>GM10-TC2</td>
</tr>
<tr>
<td>GM</td>
<td>10</td>
<td>BC- 1 Cover Plates on I/H sxn</td>
<td>GM10-BC3</td>
</tr>
</tbody>
</table>

*STAAD.Pro 3161 User Manual*
### Member Grouping System in STAAD.Pro

<table>
<thead>
<tr>
<th>Cantilever Label</th>
<th>Group Type</th>
<th>Grp Number</th>
<th>STAAD Group Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>NC</td>
<td>GS</td>
<td>1001</td>
<td>_NC-GS-1001</td>
</tr>
<tr>
<td>EC</td>
<td>GM</td>
<td>1002</td>
<td>_EC-GS-1002</td>
</tr>
<tr>
<td>SC</td>
<td>BS</td>
<td>1003</td>
<td>_SC-BS-1003</td>
</tr>
<tr>
<td>WC</td>
<td>GM</td>
<td>21</td>
<td>_WC-GM-21</td>
</tr>
<tr>
<td>C</td>
<td></td>
<td>51</td>
<td>_C-51</td>
</tr>
<tr>
<td>V</td>
<td></td>
<td>101</td>
<td>_V-101</td>
</tr>
<tr>
<td>HV</td>
<td></td>
<td>2005</td>
<td>_HV-2005</td>
</tr>
<tr>
<td>VB (Vert Bracing)</td>
<td></td>
<td>100</td>
<td>_VB-100</td>
</tr>
<tr>
<td>PST (Post)</td>
<td></td>
<td>100</td>
<td>_PST-100</td>
</tr>
<tr>
<td>PTL (Portal)</td>
<td>1</td>
<td></td>
<td>PTL-1-ISM C150-150</td>
</tr>
</tbody>
</table>

### Memb Type

<table>
<thead>
<tr>
<th>Memb Type</th>
<th>Section Ref No</th>
<th>Built Up Reference</th>
<th>Labels</th>
</tr>
</thead>
<tbody>
<tr>
<td>V</td>
<td>33</td>
<td>L - Long Legs of Angle</td>
<td>V33-L1</td>
</tr>
<tr>
<td>V</td>
<td>33</td>
<td>- Equal Angle</td>
<td>2V33-10</td>
</tr>
<tr>
<td>V</td>
<td>34</td>
<td>S - Short Legs for angle</td>
<td>V34-S2</td>
</tr>
<tr>
<td>GM</td>
<td>35</td>
<td>SA - Star of angles</td>
<td>GM35-SA1</td>
</tr>
<tr>
<td>V</td>
<td>10</td>
<td>SI - Star Of I/H Sxn</td>
<td>V10-SI2</td>
</tr>
<tr>
<td>GM</td>
<td>0 - BU</td>
<td>BU - Built Up Girder out of plates</td>
<td>GM0-BU1</td>
</tr>
<tr>
<td>GM</td>
<td>15</td>
<td>2 I/H Sections</td>
<td>2GM15-10</td>
</tr>
</tbody>
</table>
## Connection Design tab

### Table 348: Assign Connections group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Joint Cursor</td>
<td>Used to select joints for connection design.</td>
</tr>
<tr>
<td>Select Joints &gt;</td>
<td>&lt;joint types&gt; Tools for selecting joints in the model by member type and orientation.</td>
</tr>
<tr>
<td>Select Special Joints</td>
<td>Opens the Special selection of joints dialog, which is used to select joint types by category as well as by filtering for member depth and end release.</td>
</tr>
<tr>
<td>Select Connections &gt;</td>
<td>Select all connections This will select all the connection in the model</td>
</tr>
<tr>
<td>Select identical connections</td>
<td>This will select all the identical connection in the view</td>
</tr>
<tr>
<td>Tool name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Select connections of the same type</strong></td>
<td>This will select the connection of same type.</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>To use this feature, you must select at least one connection before selecting this tool.</td>
</tr>
<tr>
<td><strong>Select connections with same template</strong></td>
<td>This will select the connection of same template.</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>To use this feature, you must select at least one connection before selecting this tool.</td>
</tr>
<tr>
<td><strong>Select connections of the same family</strong></td>
<td>This will select the connection of same family name, as used in RAM Connection.</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>To use this feature, you must select at least one connection before selecting this tool.</td>
</tr>
<tr>
<td><strong>Select connections with same tag names</strong></td>
<td>This will select the connection of same tag name, as used in RAM Connection.</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>To use this feature, you must select at least one connection before selecting this tool.</td>
</tr>
<tr>
<td><strong>Select connections with same template and sections</strong></td>
<td>This will select the connection of same template and where the section of the members which participate in the connection are also same.</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>To use this feature, you must select at least one connection before selecting this tool.</td>
</tr>
<tr>
<td><strong>Select connections nodes and members</strong></td>
<td>This will select all the nodes and members which participate in the connection.</td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>To use this feature, you must select at least one connection before selecting this tool.</td>
</tr>
<tr>
<td><strong>Assign Basic Connection</strong></td>
<td>Opens the Basic Connections dialog, which is used to assign basic connection templates to selected joints.</td>
</tr>
<tr>
<td><strong>Assign Smart Connection</strong></td>
<td>Opens the Smart Connections dialog, which is used to assign smart connection templates to selected joints.</td>
</tr>
</tbody>
</table>
### Tool name: Assign Gusset Connection

**Description:**
Opens the *Gusset Connections* dialog, which is used to assign gusset connection templates to selected joints.

---

### Table 349: Frames group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| ![icon] | **Identify Beam and Grider**

**Description:**
Opens the *Beam-Girder Identification* dialog, which is used to switch the beam and girder assignments for connections.

**Note:** If no Beam-Girder (BG) connections have been assigned, a warning dialog opens to notify you no matching connections could be found.

| ![icon] | **Create Seismic Frame**

**Description:**
Opens the *Seismic Frames* dialog, which is used to specify the type of seismic frame used for the selected members creating a seismic frame definition.

---

### Table 350: Reports group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| ![icon] | **Connection Report**

**Description:**
Opens the *RAM Report Export* dialog, which is used to manage connections and other report details to include in an report.

---

### Table 351: Configure group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| ![icon] | **Connection Database**

**Description:**
Opens the connection database dialog, which is used to manage and create connection templates.
Special Selection of Joints dialog

Used to select joint types by category as well as by filtering for member depth and end release.

Opens when the Select Joints > Select Special Joints tool is selected in the Assign Connections group on the Connection Design ribbon tab.

- **Select from** Choose an option by which to limit the selecting range if you had previously selected members prior to this dialog opening.
- **Family** Select one connection family type. These are the same options as found on the Select Joints tool in the Assign Connections group on the Connection Design ribbon tab.
- **Limit selection by beam depth** Check this option and provide minimum and maximum beam depth values (in the units displayed) in order to further limit the matching connections for selection.
- **Verify Releases** Check this option and then check one or both of the end release types (pinned or fixed).

Basic Connections dialog

Used to assign Basic connection templates to selected joints. A “Basic” connection template contains all the information about the connection (such as the plate sizes and bolt locations etc) which is applied to a joint and then checked for code compliance.

A set of joints that is to use Basic connections can contain multiple connections (i.e., each with a different plate size and bolt diameter). If the first connection does not achieve compliance, then the next is checked until either a suitable connection is defined or all the selected connections have been checked. Therefore, when assigning basic connections, you will typically select multiple connections to be checked.

Opens when Connection Design > Assign Basic Connections is selected.
Note: The definition of each connection template and the order in which they occur, are defined in a database which can be displayed and/or modified from the Connection Design menu. See the RAM Connection manual for more information on editing the connection database.

**Design**

Select the design code to be used. The field is populated with the default code selected in the RAM Settings dialog.
Seismic Provisions (AISC codes only)
Select the Consider AISC 341-05 Seismic Provisions option to instruct the program to
design the connections per this specification.
A gravity load combination must be selected when this option is used.

Consider Structural Integrity (BS5950 only) Select this option to include the requirements for structural integrity in
the design (Cl 2.4.5.2 BS5950 1:2000). A tie force not less than 75 kN is considered for
these verifications. The appendices A to D (BS5950 1:2000) give the required
information on the behavior and methodology to resist tying forces in the design.

Seismic Category (Base Plate only) Specify the site class which best describes the soil conditions
supporting the foundation and base plate.

Connection types Select the class of connection to be used from this list. The connection types available
depend on the code selected.

Available list A list of all available connection templates for the selected connection type are listed here.

List Operators

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected connection template to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all connection templates to the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all entries in the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected entry from the Selected list.</td>
</tr>
</tbody>
</table>

Selected list The connection templates selected for design will be added to the list.

Design Connections Individually Select this option to instruct the program to optimize connections at each joint
separately, rather than grouping for the worst case among common connections.

Button Description

OK Accepts the design criteria and performs a design for the selected joints using the selected connection
templates.

Cancel Closes the dialog without designing any connections.

Help Opens the STAAD.Pro help window.

Related Links
- D. Design Connections Individually (on page 971)
- D. To select a basic connection template (on page 967)
Smart Connections

Used to assign smart connection templates to selected joints. A “smart” connection template contains parametric rules defined in a macro which allows characteristics of the connection to be modified (within limits) in order to achieve code compliance. A joint selection set that is to use a Smart connection can only specify a single smart connection definition.

Opens when **Connection Design** > **Assign Smart Connections** is selected.
Note: The definition of each connection template and the order in which they occur, are defined in a database which can be displayed and/or modified from the Connection Design menu. See the RAM Connection manual for more information on editing the connection database.

- **Design**: Select the design code to be used. The field is populated with the default code selected in the RAM Settings dialog.

  - **Seismic Provisions** (AISC codes only)
    - Select the Consider AISC 341-05 Seismic Provisions option to instruct the program to design the connections per this specification.
    - A gravity load combination must be selected when this option is used.

  - **Consider Structural Integrity** (BS5950 only)
    - Select this option to include the requirements for structural integrity in the design (Cl 2.4.5.2 BS5950 1:2000). A tie force not less than 75 kN is considered for these verifications. The appendices A to D (BS5950 1:2000) give the required information on the behavior and methodology to resist tying forces in the design.

  - **Seismic Category** (Base Plate only)
    - Specify the site class which best describes the soil conditions supporting the foundation and base plate.

- **Connection types**: Select the class of connection to be used from this list. The connection types available depend on the code selected.

- **Available list**: A list of all available connection templates for the selected connection type are listed here.

- **List Operators**
  - **Click this button…**
  - **to…**
    - > Add the selected connection template to the Selected list.
    - >> Add all connection templates to the Selected list.
    - << Remove all entries in the Selected list.
    - < Remove the selected entry from the Selected list.

  - **Selected list**: The connection templates selected for design will be added to the list.

- **Design Connections Individually**: Select this option to instruct the program to optimize connections at each joint separately, rather than grouping for the worst case among common connections.

- **Button Description**
  - **OK**: Accepts the design criteria and performs a design for the selected joints using the selected connection templates.
  - **Cancel**: Closes the dialog without designing any connections.
  - **Help**: Opens the STAAD.Pro help window.
Ribbon Control Reference
Connection Design tab

Related Links
- D. Design Connections Individually (on page 971)
- D. To Select a Smart Connection Template (on page 968)

**Gusset Connections** dialog

Used to assign gusset connection templates to selected joints. A Gusset connection template contains parametric rules similar to those in a Smart connection template but that are limited to connections using gusset plates.

Opens when Connection Design > Assign Gusset Connections is selected.
Note: The definition of each connection template and the order in which they occur, are defined in a database which can be displayed and/or modified from the **Connection Design** menu. See the RAM Connection manual for more information on editing the connection database.

**Design**
Select the design code to be used. The field is populated with the default code selected in the RAM Settings dialog.

**Seismic Provisions**  (AISC codes only)

Select the Consider AISC 341-05 Seismic Provisions option to instruct the program to design the connections per this specification.

A gravity load combination must be selected when this option is used.

**Consider Structural Integrity** (BS5950 only) Select this option to include the requirements for structural integrity in the design (Cl 2.4.5.2 BS5950 1:2000). A tie force not less than 75 kN is considered for these verifications. The appendices A to D (BS5950 1:2000) give the required information on the behavior and methodology to resist tying forces in the design.

**Seismic Category** (Base Plate only) Specify the site class which best describes the soil conditions supporting the foundation and base plate.

**Connection types** Select the class of connection to be used from this list. The connection types available depend on the code selected.

**Available list** A list of all available connection templates for the selected connection type are listed here.

**List Operators**

<table>
<thead>
<tr>
<th>Click this button…</th>
<th>to…</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt;</td>
<td>Add the selected connection template to the Selected list.</td>
</tr>
<tr>
<td>&gt;&gt;</td>
<td>Add all connection templates to the Selected list.</td>
</tr>
<tr>
<td>&lt;&lt;</td>
<td>Remove all entries in the Selected list.</td>
</tr>
<tr>
<td>&lt;</td>
<td>Remove the selected entry from the Selected list.</td>
</tr>
</tbody>
</table>

**Selected list** The connection templates selected for design will be added to the list.

**Design Connections Individually** Select this option to instruct the program to optimize connections at each joint separately, rather than grouping for the worst case among common connections.

**Button Description**

**OK** Accepts the design criteria and performs a design for the selected joints using the selected connection templates.

**Cancel** Closes the dialog without designing any connections.

**Help** Opens the STAAD.Pro help window.

**Related Links**

- *D. Design Connections Individually* (on page 971)
- *D. To Select a Gusset Connection Template* (on page 968)
Beam-Girder Identification dialog

Used to switch beam and girder assignments in a Beam-Girder connection. STAAD.Pro will automatically assign the Beam and Girder designations. This dialog allows you to switch the assignment between the two members in the connection.

Opens when Connections > Identify Beam and Girder is selected.

Table of BG Connections
All Beam-Girder (BG) connections in the model are listed here along with the members in the connection which are have been automatically assigned as Beam or Girder. Select the Check option for each connection for which you wish to switch member assignments.

Switch
Click this button to swap the Beam No and Girder No member assignments of members selected in the Check column.

OK
Accepts the settings changes and closes the dialog.

Cancel
Closes the dialog without saving changes.
Help

Opens the Help window.

Seismic Frames

Used to specify the type of seismic frame used for the selected members creating a seismic frame definition.

Note: This dialog is accessed from the [Connection | Seismic Frame page](on page 979).

Opens when **Connection > Create Seismic Frame** is selected.

The `Seismic Frames` dialog allows you to select the predefined method of lateral seismic force resisting system the frame represents. The options include:

- None
- Ordinary Moment Frames (OMF)
- Intermediate Moment Frames (IMF)
- Special Moment Frames (SMF)
- Ordinary Concentrically Braced Frames (OCBF)
- Special Concentrically Braced Frames (SCBF)

**OK**

Accept the seismic frame assignment and closes the dialog.

**Cancel**

Closes the dialog without making a seismic frame assignment.

RAM Report Export dialog

Used to manage connections and other report details to include in an report.

Note: The report is saved to a Microsoft Office Word document in the same folder as the STAAD.Pro input file, as `inputfile_Report.doc`, where `inputfile` is the STAAD.Pro Input filename.

- **Connection list**
  Contains a list of all the designed connections. Use the Up and Down buttons to reorder the connections in the report. Select the connections you want to include in the report.

- **Data Report**
  Check this option to include the input values in the report.

- **Result Report**
  Set this option to include the code check output for the connections in the report.

- **Formula**
  Check this option to include the formulas used for code checks in the report.

- **Individual / Merged Report**
  Select to generate either individual reports for each connection or a single report with all connections merged into a single file.
Related Links

- D. To Export Connection Designs to a Report (on page 971)

**RAM Connection Material Database** dialog

Used to review additional material data required by Connection Design for the design of steel connections. Material strengths are displayed here to allow for the assignment of material grades.

Opens when **Connection Design > Show RAM Database** is selected.

Additional material data is required by RAM Connection beyond that which is kept in STAAD.Pro's databases. This information can be viewed and edited from the RAM Material dialog. In the case of the same material being defined in both STAAD and RAM Connection, the local STAAD value is used.

The STAAD materials are read from the STAAD.Pro .ini file. Some generic materials do not have any strength materials defined and these should be entered here.

Materials listed in the country tabs (i.e., United States or United Kingdom) are read from RAM Connection. These values are displayed in yellow fields and may not be edited in this dialog.

**Materials table**  The following explains the columns used in the RAM Materials table:
### Material Property

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fy</td>
<td>Yield stress</td>
</tr>
<tr>
<td>Ry</td>
<td>Yield strength ratio</td>
</tr>
<tr>
<td>Rt</td>
<td>Ratio of the expected tensile strength to the specified minimum tensile strength, Fu</td>
</tr>
<tr>
<td>Fu</td>
<td>Specified minimum tensile strength</td>
</tr>
<tr>
<td>FCu</td>
<td>Ultimate strength of concrete</td>
</tr>
</tbody>
</table>

**OK**
Accepts the material changes entered in the table and closes the dialog.

**Cancel**
Closes the dialog without saving changes.

**Help**
Opens the Help window.

### Bolts tab
Database tables lists bolt geometry and material properties. The following tables are included:

- **AISC** - American Institute of Steel Construction bolts, in imperial units.
  - Db: Bolt Diameter
  - Fv: Nominal shear stress
  - Ft: Nominal tensile stress
  - Tm: Minimum Bolt pretension
  - Mu: Minimum Slip Coefficient
- **BS** - British Standard bolts, in metric.
  - Db: Bolt Diameter
  - Pt: Tensile Strength
  - Ps: Shear Strength
  - P0: Minimum Shank Tension
  - Mu: Slip Factor

### Weld tab
Database table lists weld material properties. The following tables are included:

- **AISC** - American Institute of Steel Construction welds, in imperial units.
  - Fexx: Weld electrode Strength

### Advanced Slab Design tab
Contains tools used for selecting elements of a defined slab as well as for transmitting data to RAM Concept for slab design.
### Table 352: Highlight group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Highlight Plates</td>
<td>Selects all plate elements associated with the selected slab definition in the Slabs table.</td>
</tr>
<tr>
<td>Highlight Beams</td>
<td>Selects all beam members associated with the selected slab definition in the Slabs table.</td>
</tr>
<tr>
<td>Highlight Columns</td>
<td>Selects all column members associated with the selected slab definition in the Slabs table.</td>
</tr>
<tr>
<td>Highlight Geometry</td>
<td>Selects all structural elements associated with the selected slab definition in the Slabs table.</td>
</tr>
</tbody>
</table>

### Table 353: RAM Concept group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Export to RAM Concept</td>
<td>Creates the required data files for RAM Concept to analyze and design the selected slab definition in the Slabs table.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> This command does not open the external slab design program nor does it execute the analysis and design functions in that program.</td>
</tr>
<tr>
<td>Open in RAM Concept</td>
<td>Creates the required data files for RAM Concept to analyze and design the selected slab definition in the Slabs table. RAM Concept is launched.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If RAM Concept is not installed on your computer or if the version installed is not compatible with Advanced Slab Design link, a warning dialog opens to inform you that the data could not be sent to RAM Concept.</td>
</tr>
</tbody>
</table>
**Node Tools tab**

*Note:* The Clipboard group contains **Cut**, **Copy**, **Paste**, and **Delete** tools found on the **Geometry** ribbon tab.

### Table 354: Model group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Move Joint Icon]</td>
<td>Opens the <strong>Move Entities</strong> dialog, which is used to specify the translational offset for moving a selection of nodes.</td>
</tr>
<tr>
<td>![Renumber Nodes Icon]</td>
<td>Opens the <strong>Renumber</strong> dialog, which is used to renumber selected nodes starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.</td>
</tr>
<tr>
<td>![Assign Supports Icon]</td>
<td>Opens the <strong>Supports - Whole Structure</strong> dialog, which is used to define supports and assign them to nodes.</td>
</tr>
</tbody>
</table>

### Table 355: View group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>![New View Icon]</td>
<td>Opens the <strong>New View</strong> dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way.</td>
</tr>
</tbody>
</table>

*Note:* This option becomes active only after you select one or more structural elements on screen.
### Tool name | Description
---|---
3D Rendering | Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view. 

Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options. 

Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

#### Node dialog

Displays information for the selected node. 

Opens when a node is double clicked with the Nodes Cursor.

**Node**

Displays the node number and coordinates for the currently selected node.

**Nodes**

Opens the [Nodes table](on page 729).

**Supports**

Opens the [Supports table](on page 730).

**Loads**

Opens the [Load Values table].

**Reactions**

Opens the [Support Reactions table](on page 2225).

**Displacements**

Opens the [Node Displacements table](on page 2225).

If no results have been setup, the [Results Setup dialog](on page 3147) will be opened first to specify the results you wish to view.

**Close**

Closes the dialog.

**Related Links**

- [GS. Object Properties Inspection](on page 49)

---

**Beam Tools tab**

*Note:* The Clipboard group contains **Cut**, **Copy**, **Paste**, and **Delete** tools found on the **Geometry** ribbon tab.
### Table 356: Model group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Move Beams</td>
<td>Opens the <strong>Move Selected Beams</strong> dialog, which is used to specify the translational offset for moving a selection of beams.</td>
</tr>
<tr>
<td>Insert Node</td>
<td>Opens the <strong>Insert Node into Beam #</strong> dialog, which is used to insert one or more nodes at specified distances along selected members.</td>
</tr>
<tr>
<td>Form Member</td>
<td>Used to manually form a physical structural member from a selection of one or more connected analytical beam segments.</td>
</tr>
<tr>
<td>Stretch Beam</td>
<td>Opens the <strong>Stretch Member(s)</strong> dialog, which is used to increase the length of a member in various ways.</td>
</tr>
<tr>
<td>Merge Beams</td>
<td>Opens the <strong>Merge Selected Beams</strong> dialog, which is used to join two collinear beams and replace them with one beam.</td>
</tr>
<tr>
<td>Intersect Beams</td>
<td>Splits members at detected intersection points and creates an addition node at this intersection point such that the members are now connected.</td>
</tr>
<tr>
<td>Renumber Beams</td>
<td>Opens the <strong>Renumber</strong> dialog, which is used to renumber selected beams starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.</td>
</tr>
<tr>
<td>Define Section Profile</td>
<td></td>
</tr>
</tbody>
</table>
### Tool name | Description
---|---
| Properties | Opens the **Beam** dialog which displays properties for the selected member. Additional tabs become available after analysis and design is performed.

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assign</td>
<td>Opens the <strong>Assign Connection Tags</strong> dialog and <strong>New Connection tag</strong> dialog, which are used to assign existing connection tags to member ends in the model and to create new connection tags, respectively.</td>
</tr>
<tr>
<td>Remove</td>
<td>Opens the <strong>Assign Connection Tags</strong> dialog and initiates the remove connection tag tool for the connections on the selected member. A confirmation dialog opens to confirm the remove action.</td>
</tr>
<tr>
<td>View</td>
<td>Opens the <strong>Assign Connection Tags</strong> dialog.</td>
</tr>
<tr>
<td>Check</td>
<td>(Active only after a successful analysis has been performed) Opens the <strong>Assign Connection Tags</strong> dialog and <strong>Check Connection Tags</strong> dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.</td>
</tr>
</tbody>
</table>

### Table 357: Connection Tags group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>New View</td>
<td>Opens the <strong>New View</strong> dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way. <strong>Note:</strong> This option becomes active only after you select one or more structural elements on screen.</td>
</tr>
</tbody>
</table>
### Tool name

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| **3D Rendering**    | Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view.  
Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options.  
Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only. |

### Plugins

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| **Add Attribute**    | Opens the Member Attribute dialog which is used to select and apply a member attribute to selected members.  
Refer to TR.29.1 Struclink Member Attribute (on page 2536) for additional information on the Struclink attribute which is assigned using this dialog. |
| **List/Delete Attribute** | Opens the Member Attribute dialog to display the currently used Member Attribute for a single selected member                                                                                                                                                                                                  |

### Assign Connection Tags dialog

Used to assign existing connection tags to member ends in the model.

Opens when Connection Tags > *any* Connection Tags is selected from the right-click pop-up menu when a member is selected.
Table 359: Dialog toolbar controls

<table>
<thead>
<tr>
<th>Toolbar icon</th>
<th>What It Does</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="New" /></td>
<td>Opens the New Connection Tag dialog, which is used to define new connection tags and optionally assign member end restraints associated with a connection tag.</td>
</tr>
<tr>
<td><img src="image" alt="Delete" /></td>
<td>Used to remove the selected connection tag entry from the table. A confirmation dialog opens to confirm you want to remove the entry.</td>
</tr>
</tbody>
</table>
Check
(Disabled if no current analysis results are available for the current STAAD.Pro model) Opens the Check Connection Tags dialog, which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

Each row in the table is a separate connection tag added to the model from the Connection Tags XML file.

Select one of the following methods for assigning connection tags:

- **Assign to Selected Beams** — Uses the current members selected in the main view window
- **Assign to View** — Uses all members visible in the main view window
- **Assign to Edit List** — Uses the member numbers listed in the associated text field

> Tip: You can type a dash between two numbers to indicate all member numbers, inclusive.

Assigns the currently selected connection tags in the table using the selected Assignment Method. The members assigned are added to the Assigned Beams cell in the row.

Closes the dialog.

**Related Links**

- *D. To create a connection tag* (on page 982)

**New Connection Tag** dialog

Used to define new connection tags and optionally assign member end restraints associated with a connection tag.

Opens when either:

- **Connection Tags > Remove Connection Tags** is selected from the right-click pop-up menu when a member is selected, or
- the **New** tool is selected in the Assign Connection Tags dialog
**Figure 345: The New Connection Tag dialog**

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Select Categories</strong></td>
<td>Use the drop down list to select one of the connection tag categories defined in the Connection Tag XML file. Additional category descriptions are displayed.</td>
</tr>
<tr>
<td><strong>Select Tags</strong></td>
<td>Use the drop down list to select one of the connection tags defined in the selected category. When selected, the member end releases defined for this connection tag are indicated in the inactive degree of freedom check boxes in the Releases group.</td>
</tr>
<tr>
<td><strong>Assign Beam End Releases</strong></td>
<td>Set this check box option on to assign the indicated member end releases for the connection tag in the STAAD.Pro model when the connection tag is assigned.</td>
</tr>
<tr>
<td><strong>Tip</strong></td>
<td>This is recommended so the releases match those assumed by the connection tag.</td>
</tr>
<tr>
<td><strong>Location</strong></td>
<td>Select either the <strong>Start</strong> or <strong>End</strong> member end (or both) for adding this connection tag to the Assign Connection Tags dialog table or assigning to currently selected members.</td>
</tr>
<tr>
<td><strong>Note</strong></td>
<td>At least one Location must be selected in order to add or assign the connection tag.</td>
</tr>
<tr>
<td><strong>Start</strong></td>
<td>The start node of the member.</td>
</tr>
<tr>
<td><strong>End</strong></td>
<td>The end node of the member.</td>
</tr>
<tr>
<td><strong>Add</strong></td>
<td>Adds the selected connection tags at the selected location to the Assign Connection Tags dialog table.</td>
</tr>
<tr>
<td><strong>Close</strong></td>
<td>Closes the dialog.</td>
</tr>
<tr>
<td><strong>Assign</strong></td>
<td>Assigns the current connection tags to the current member selection in the STAAD.Pro model. These connection tags will also be added to the Assign Connection Tags dialog.</td>
</tr>
</tbody>
</table>

**Related Links**
- [D. To create a connection tag](#) (on page 982)
Remove Connection Tags dialog

Used to select which member end from which the connection tag assignment will be removed.

Opens when **Connection Tags > Remove Connection Tags** is selected from the right-click pop-up menu when a member is selected.

![Remove Connection Tag dialog](image)

*Figure 346: The Remove Connection Tag dialog*

- **Location**
  Select one or both of the **Start** or **End** member ends for removing a connection tag assignment.

- **Remove Beam End Releases**
  Set this option to have the associated member end releases returned to default (fixed) in the STAAD.Pro input file.

- **Remove**
  Performs the assignment removal. A confirmation dialog opens

- **Close**
  Closes the dialog without removing any connection tag assignments.

**Related Links**

- [D. To remove connection tag assignments](on page 984)

Beam dialog

The member query box includes the ability to change many of the member attributes from the dialog box itself including the property definition, specifications (releases, truss, cable, etc.) and beta angle.

A list of the load items assigned to the member are displayed here. New load items can now be added to existing load cases, and, existing load items edited or deleted using the Member Query window.

- **Select Load Case**
  Choose the load case to you want to display or edit. This list will be empty if no primary load cases have been defined in the model.

- **Add new Load Item**
  Opens the Create New Load Items dialog, which is used to add new load items to the selected load case and assign them to the currently selected member. The list of load items displayed is limited to those which can be assigned to members.

- **Remove Selected Load Item**
  Click to remove an existing load item selected in the list.

- **Edit Selected Load Item**
  Click to make changes to the currently selected load item in the list. The load item dialog for the load type opens. Make the desired changes and then click **Change**.

Opens when a member is double-clicked with the Beams cursor.
Tip: While this dialog box is open, you may select any other member to Query by double-clicking it. This updates
the dialog box contents automatically for the newly selected member. Also, the units can be changed dynamically
while this dialog box is open. Changing the units using the View | Options or Tools | set Current Display Units
menu affects the contents of this dialog box automatically.

To change the property, member specifications or the beta angle on the member, simply click on one of the
buttons. If the member contains output result tabs (Displacement, Steel Design, etc.) in the query box, changing
the member attributes will cause these tabs to disappear. This is due to the fact that the current output no longer
reflects the new input.

Note: Changing the member attributes for one member will subsequently change the attributes of all other
members belonging to the same attribute list. For example, if the current member’s property is also assigned to
other members, changing the property on the current member will change the property on all the members.

Geometry
Shows a diagram of the member property and lists the length, connecting nodes and their coordinates, and
whether the member has a beta angle, any releases or other member specifications. One can assign/change the
beta angle of the member by clicking on the Change Beta button.

<table>
<thead>
<tr>
<th>Change Beta</th>
<th>Click to assign or change the beta angle for the selected member.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Change Releases</td>
<td>Click to assign or change the start or end release attributes, member offsets, cable</td>
</tr>
<tr>
<td></td>
<td>specification, truss specification, tension-only specification, compression-only</td>
</tr>
<tr>
<td>At Start / End</td>
<td>specification, ignore stiffness, or inactive specifications.</td>
</tr>
</tbody>
</table>

Property
Shows a diagram of the member property and lists the length, cross sectional and material properties. One can
assign/change the property or member specifications of the member by clicking on the respective buttons.

| Assign/Change     | Click to assign or change the section property of the member. The choice of properties will               |
| Property          | only from the same family as the current cross-section.                                                |
|                   | For example, if the current property is an America W12x45, only W, M, S, HP, and B shapes               |
|                   | and sizes will be available.                                                                           |

| Assign Material   | Click to update the material definition with one selected from the drop-down list.                     |

Shear Bending
This tab appears only after running the analysis. It also contains facilities for viewing values for shears and
moments, selecting the load cases for which those results are presented, a slider bar for looking at the values at
specific points along the member length, and a Print option for printing the items on display.

The slider bar in the member query box can be used to display the shear, bending and deflection values at any
point along the beam. A corresponding edit box is also provided to manually type in the distance for which the
aforementioned values will be calculated and displayed.

To obtain the values for the shear, bending or deflection at any point along the beam for a specific load case, grab
the slider bar (the draggable arrow) and drag it to the desired location on the beam. The location of the slider
bar is reported in the Dist edit box in the current display units. This edit box can also be manually changed which
conversely, changes the location of the slider bar. The values for the results in question will automatically be
updated every time the slider bar moves.
Deflection
This tab appears only after running the analysis. Shows deflection diagram in the global or local directions for the selected load case. Also lists tabulated deflection values. Points of inflection are displayed on the diagram. The points of inflection or contraflexure are the distances along the member where the sign of the force, deflection or moment changes (the point where these values are zero).
Refer to the next figure to see where the points of inflection are labeled. The distances are always reported from the start of the member (in the figure, the left-hand side of the member).

Composite Property
The dialog box displays various properties for the composite beam including effective width, whether shoring during construction has been considered or not, grade of concrete, unit weight of concrete, thickness of concrete above flutes, rib properties, stud properties etc. These data will enable the user to get a clear picture of the geometric properties for the composite beam.

Design Property
If the STAAD input file contains instructions for steel design - Member Selection (See section 5.47.3 of the STAAD Technical Reference Manual, and Example problem 1 of the examples manual) which may result in a change of section property from the one originally assigned by the user - that new section name, and associated property values will be displayed under this tab. One can assign/change the material attributes for the beam by clicking on the Assign Material button.

Steel Design
This tab appears only after running the analysis if a steel design operation has been performed on the selected member. Shows a diagram of the member property and lists the length, design stresses, critical load, code being used, ratio, result, critical condition and slenderness value.

\begin{itemize}
  \item \textbf{fSB} \hspace{1cm} \text{Actual Steel Stress before Concrete Hardens}
  \item \textbf{FSB} \hspace{1cm} \text{Allowable Steel Stress before Concrete Hardens}
  \item \textbf{fCB} \hspace{1cm} \text{Actual Concrete Stress before Hardening}
  \item \textbf{FCB} \hspace{1cm} \text{Allowable Concrete Stress before Hardening}
  \item \textbf{fSA} \hspace{1cm} \text{Actual Steel Stress after Concrete Hardens}
  \item \textbf{FSA} \hspace{1cm} \text{Allowable Steel Stress after Concrete Hardens}
  \item \textbf{fCA} \hspace{1cm} \text{Actual Concrete Stress after Hardening}
  \item \textbf{FCA} \hspace{1cm} \text{Allowable Concrete Stress after Hardening}
\end{itemize}

\textbf{Critical load} \hspace{1cm} The table displays information like critical load case, critical force values, and location of the critical section along the member length

\textbf{Code} \hspace{1cm} The design code followed for designing the composite beam
\textbf{Result} \hspace{1cm} Displays whether the beam has PASSED or FAILED the check
\textbf{Ratio} \hspace{1cm} Ratio of actual to allowable stresses
\textbf{Critical} \hspace{1cm} Reports the critical condition. In the previous dialog box CONC-STR-AH is displayed which indicates that Concrete Stress after hardening is the critical condition.
Display the number of shear connectors required and diameter of shear connectors.

Castellated Beam Design

Concrete Design

Related Links

- GS. Object Properties Inspection (on page 49)

Physical Member dialog

Used to display information about a selected physical member, similar to the member query dialog.

Related Links

- GS. Object Properties Inspection (on page 49)

Plate Tools tab

This ribbon tab is available when one or more plate objects are selected.

Note: The Clipboard group contains Cut, Copy, Paste, and Delete tools found on the Geometry ribbon tab.

Table 360: Model group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Image] Move Plate</td>
<td>Opens the Move Selected Plates dialog, which is used to specify the translational offset for moving a selection of plates.</td>
</tr>
<tr>
<td>![Image] Renumber Plates</td>
<td>Opens the Renumber dialog, which is used to renumber selected plates starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.</td>
</tr>
<tr>
<td>![Image] Generate Plate Mesh</td>
<td>Used to generate a finite element mesh for an existing plate element. The polygon can be meshed into quadrilateral or triangular elements. You have control over parameters like number of divisions along each side of the polygon. Polygonal holes can also be defined within the surface during the meshing process for polygonal meshing.</td>
</tr>
</tbody>
</table>
### Plate Tools tab

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Plate" /> Properties</td>
<td>Opens the <strong>Plate</strong> dialog which displays properties for the selected element. Additional tabs become available after analysis and design is performed.</td>
</tr>
</tbody>
</table>

#### Table 361: View group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| ![New View](image) New View | Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of "child" view windows in this way.  

**Note:** This option becomes active only after you select one or more structural elements on screen. |
| ![3D Rendering](image) 3D Rendering | Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view.  

Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options.  

Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only. |

---

**Plate** dialog

Used to view a summary of the input and output data for a plate element, including geometry, properties, and output results.

If the structure has been successfully analyzed, there will be three tabs on the plate query box: Geometry, Property Constants and Stresses. If there are no analysis results, the last tab (Stresses) will not be shown.

Opens when a quad or triangular plate element is double-clicked with the Plates cursor.

**Geometry tab**

Displays the plate incidences, the plate corner coordinates and the lengths of each of the edges.
There are certain attributes which can be assigned to plates in order to simulate characteristics like the type of connections that are present at their corners. These are - Releases of individual degrees of freedom at its nodes, defining them as Plane Stress elements, ignoring their stiffness for the purpose of analysis, etc. If such attributes are assigned, those will be listed against the Plate Spec: field.

Property Constants tab

Displays the thickness of each of the sides of the plate and the material properties. One may change/assign the properties of the plate by clicking on the Assign/Change Property button. One can also assign/change the material attributes by clicking on the Assign Material button.

Center Stresses tab

Displays the 8 center stresses along with the top and bottom Principal, Von Mises and Tresca stresses for a particular load case. The load case can be changed using the Load List dropdown box.

Corner Stresses tab

Used to display the stresses at the corner node points of the selected plate. The node numbers of the selected plate and the stresses at those nodes are displayed.

Load List
Select any one of the predefined load cases for displaying the corner stresses of the selected plate due to that particular type of loading.

Plate Corner Stresses
Different types of stresses at the corner nodal points of the selected plate under the specified load case are listed here.

Princ Stress and Disp tab

Used to display the nodal displacements at corner points of the selected plate.

Load List
Select any one of the predefined load cases for displaying the displacements at the corner node points and also the principal stresses of the selected plate due to that particular type of loading.

Plate Corner Displacement
After selecting a particular type of load case in the Load List, displacements in the X, Y, and Z direction at the corner node points of the selected plate will be listed here.

Plate Principal Stresses
Top and bottom principal stresses of the selected plate under the specified load case are displayed here.

Related Links
• GS. Object Properties Inspection (on page 49)

Surface Query dialog

Used to view detailed information for one surface entity at a time.

Opens when a surface element is double-clicked with the Surface cursor.

While this dialog box is open, you can query any other surface element by double-clicking on it. The dialog updates automatically for the newly selected element. Similarly, the display units can be changed dynamically while this dialog box is open using the Change Graphical Display dialog.
Geometry tab
Shows a diagram of the surface element and a table listing all nodes on the perimeter of the element, as well as node to node distances along the perimeter.

Property tab
Lists surface element’s thickness and properties of the element’s material: Elasticity, Poisson’s ratio, Density, and thermal expansion coefficient (Alpha).

Surface Forces tab
(Available after a successful shearwall analysis and design) Reports internal forces (Fx, Fy, Fxy, Mx, My, Mz, Qx, and Qy) for five levels at which the shear wall design is carried out: 1/8h, 1/4h, 1/2h, 3/4h, and 7/8h. The forces may be viewed for all load cases / combinations included in the design.

Concrete Design (Shear wall) tab
(Available after a successful shearwall analysis and design) This page of the dialog box contains a full summary of the design, including shear wall geometry, concrete and steel strengths, min. reinforcing ratios, and reinforcing details for all of the five design levels. In addition, a diagram of the horizontal cross-section of the wall is shown for each level.

Related Links
• GS. Object Properties Inspection (on page 49)

Solid Tools tab
This ribbon tab is available when one or more solid objects are selected.

Note: The Clipboard group contains Cut, Copy, Paste, and Delete tools found on the Geometry ribbon tab.

Table 362: Model group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Move Solid" /></td>
<td>Opens the Move Selected Solids dialog, which is used to specify the translational offset for moving a selection of solids.</td>
</tr>
<tr>
<td><img src="image" alt="123" /></td>
<td>Opens the Renumber dialog, which is used to renumber selected solids starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.</td>
</tr>
</tbody>
</table>
Table 363: View group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
</table>
| **New View** | Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way.  

**Note:** This option becomes active only after you select one or more structural elements on screen. |
| **3D Rendering** | Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view.  

Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options.  

Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only. |

**Solid dialog**

Similar to the beam and plate query dialog box, this will enable the users to obtain information regarding only the geometry of the solid element. As in the case of beams or plates, the user can simply double click on the solid element to see the element node numbers and the global X, Y and Z coordinates of the boundary nodes without having to scan through the Nodes table.

Opens when a solid element is double-clicked with the Solids cursor.

The table in the query box (which is not editable) displays the solid node numbers and the corresponding global X, Y and Z coordinates for the nodes.

**Related Links**
- *GS. Object Properties Inspection* (on page 49)
The examples within this section are used to validate the accuracy of the STAAD.Pro analysis and design engine.

V. Notes on Comparisons

In each verification problem, a table is used to summarize the numerical outputs from STAAD.Pro with a reference or hand calculations. A difference column presents the percent deviation from the reference, when a significant difference exists.

Where a non-significant difference (typically taken to be less than 0.5%) is present, the difference is given is “negligible.” Otherwise, the percent difference in results is listed. Where a difference of greater than 2% exists, a comment is provided regarding the difference.

This level of significance is intended to reflect that small differences in rounding or calculation methods do not present a realistic difference in the calculated result.

All verification problems have been run using the STAAD.Pro Advanced Solver. Where a numerical difference is present in the highlighted values between the Advanced and Basic Solvers, the Basic solver results are also presented for comparison.

V.01 Beams

V. Deflection and Reactions in a Beam

To find the deflection and support reactions due to a trapezoidally varying load applied on part of the span of a pinned-fixed beam.

Reference

Hand calculation using the following reference:


Problem

The beam in the following geometric, load, and section properties: $a = 3 \, \text{m}$, $b = 4.5 \, \text{m}$, $w_a = 4 \, \text{KN/m}$, $w_l = 7 \, \text{KN/m}$, $I_Z = 5,000 \, \text{cm}^4$, $E = 200 \, \text{KN/mm}^2$. 
Theoretical Solution

\[
R_A = \frac{W_a}{8l^3}(l - a)(3l + a) + \frac{W_l - W_a}{40l^3}(4l + a)
\]

\[
O_A = -\frac{W_a}{48EI}(l - a)(l + 3a) - \frac{W_l - W_a}{240EI}(l - a)^3(2l + 3a)
\]

\[
R_B = \frac{W_a - W_l}{2}(l - a) - R_A
\]

\[
M_B = R_A l - \frac{W_a}{2}(l - a)^2 - \frac{W_l - W_a}{6}(l - a)^2
\]

where

\[
l = a + b
\]

Comparison

Table 364: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotation at A, (O_A)</td>
<td>0.0020</td>
<td>0.0020</td>
<td>none</td>
</tr>
<tr>
<td>(radians)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vertical reaction at A, (R_A)</td>
<td>3.2886</td>
<td>3.2886</td>
<td>none</td>
</tr>
<tr>
<td>(kN)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vertical reaction at B, (R_B)</td>
<td>21.461</td>
<td>21.461</td>
<td>none</td>
</tr>
<tr>
<td>(kN)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Moment at B, (M_B) (kN·m)</td>
<td>25.9605</td>
<td>25.9605</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: In the STAAD model, two load cases are used. In case 1, the load is applied using the MEMBER LOAD - TRAP option. In case 2, the load is applied using a combination of MEMBER LOAD - UNI and MEMBER LOAD - LIN options. Both cases yield identical results.
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Deflection and Reactions in a Beam.STD is typically installed with the program.

STAAD PLANE REACTIONS AND DISPLACEMENTS OF A PINNED-FIXED BEAM

START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE : ROARK’S FORMULAS FOR STRESS & STRAIN
* WARREN C. YOUNG, 6TH EDITION, MCGRAW-HILL

* TABLE 3, CASE (2C), LOAD ON PARTIAL SPAN

* UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 7.5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
UNIT CM KN
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 50 IZ 5000
UNIT METER NEWTON
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+11
POISSON 0.3
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
3 FIXED
LOAD 1
MEMBER LOAD
2 TRAP GY -4 -7
LOAD 2
MEMBER LOAD
2 UNI GY -4
2 LIN GY 0 -3
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
UNIT METER NEWTON
PRINT SUPPORT REACTION
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>JOINT DISPLACEMENT (CM  RADIANS)</th>
<th>STRUCTURE TYPE = PLANE</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD  X-TRANS  Y-TRANS  Z-TRANS  X-ROTAN  Y-ROTAN  Z-ROTAN</td>
<td></td>
</tr>
</tbody>
</table>
V. Thermal Loading on a Beam

To find the support reactions due to a temperature loads applied on a fixed-fixed beam.

Reference

Hand calculation using the following reference:


Problem

The beam in the following geometric, load, and section properties: \( L = 7.5 \text{ m}, T = 40^\circ \text{F}, T_2 = 50^\circ \text{F}, \alpha = 11.7(10)^{-6}/\text{F}, d = 30 \text{ cm}, I_z = 5,000 \text{ cm}^4, E = 200 \text{ KN/mm}^2.\)
Theoretical Solution

Horizontal reactions due to case 1 loads:

\[ R_A = - R_B = EA\alpha\Delta T = [200(10)^6] \cdot [50(10)^{-4}] \cdot [11.7(10)^{-6}] \cdot (40) = 468 \text{ kN} \]

Moment reactions due to case 2 loads:

\[ M_A = - M_B = \frac{aEI\Delta T}{d} = \left[ \frac{11.7(10)^{-6}}{0.3} \right] \cdot \left[ \frac{200(10)^{-6}}{5,000(10)^{-8}} \right] = 19.5 \text{ kN} \cdot \text{m} \]

Comparison

Table 365: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal Reaction at Node A (kN)</td>
<td>468</td>
<td>468</td>
<td>none</td>
</tr>
<tr>
<td>Moment at Node A (kN·m)</td>
<td>19.5</td>
<td>19.5</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: In the STAAD model, two load cases are used. In case 1, the uniform expansion is applied. In case 2, the temperature change between top and bottom flanges is applied.
### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Thermal Loading on a Beam.STD is typically installed with the program.

STAAD PLANE TEMPERATURE LOAD ON A FIXED-FIXED BEAM

**START JOB INFORMATION**

ENGINEER DATE 18-Sep-18

**END JOB INFORMATION**

* * REFERENCE : MATRIX ANALYSIS OF FRAMED STRUCTURES
* GERE & WEAVER, 3RD EDITION, VAN NOSTRAND REINHOLD
* * TABLE B-2, APPENDIX B, PAGE 500

* * UNIT METER KN

JOINT COORDINATES

1 0 0 0; 2 3 0 0; 3 7.5 0 0;

MEMBER INCIDENCES

1 1 2; 2 2 3;

UNIT CM KN

MEMBER PROPERTY AMERICAN

1 2 PRIS AX 50 IZ 5000 YD 30

UNIT METER KN

DEFINE MATERIAL START

ISOTROPIC MATERIAL1

E 2e+08

POISSON 0.3

ALPHA 1.17e-05

END DEFINE MATERIAL

CONSTANTS

MATERIAL MATERIAL1 ALL

SUPPORTS

1 FIXED

3 FIXED

LOAD 1

TEMPERATURE LOAD

1 2 TEMP 40

LOAD 2

TEMPERATURE LOAD

1 2 TEMP 0 50

PERFORM ANALYSIS

PRINT SUPPORT REACTION

FINISH

### STAAD Output

| SUPPORT REACTIONS -UNIT KN |  |  | STRUCTURE TYPE = PLANE |
|---------------------------|--|--|--|---|--|--|--|--|
| JOINT | LOAD | FORCE-X | FORCE-Y | FORCE-Z | MOM-X | MOM-Y | MOM-Z |
| 1  | 1 | 468.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| 2 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | -19.50 |
V. Twist in a Tapered Tube

To find the twist at the free end of a hollow tapered shaft of uniform thickness.

Reference

Hand calculation using the following reference:


Problem

The beam in the following figure which has the following geometric, load, and section properties: \( L = 2 \, m \), outside diameter at fixed end = 80 mm, outside diameter at free end = 40 mm, uniform wall thickness of 10 mm, \( I_Z = 5,000 \, cm^4 \), \( E = 200 \, KN/mm^2 \), \( T = 2.0 \, KNm \), and Poisson’s ratio = 0.3.
Theoretical Solution

According to the reference, the twist at the free end is

$$\Phi_A = \left( \frac{TL}{4EI} \right) \left( \frac{C_A + C_B}{C_A^2 C_B^2} \right)$$
where

\[ C_A = \text{centerline radius at free end} = \frac{40 \text{ mm}}{2} - \frac{10 \text{ mm}}{2} = 15 \text{ mm} \]

\[ C_B = \text{centerline radius at fixed end} = \frac{80 \text{ mm}}{2} - \frac{10 \text{ mm}}{2} = 35 \text{ mm} \]

\[ G = \frac{E}{2(1 + \nu)} = \frac{200 \text{kN/m}^2}{2(1 + 0.3)} = 76.9 \]

\[ \Phi_A = \left( \frac{2 \times 200 \text{kN} \cdot \text{m} \times 2 \times 0.000 \text{m}}{4 \pi \times 76.9 \text{kN/m}^2 \times 10 \text{mm}} \right) \left( \frac{\left(15 \text{mm} + 35 \text{mm}\right)}{\left(15 \text{mm}\right)^2 \left(35 \text{mm}\right)^2} \right) = 0.0751 \text{ rad.} \]

**Comparison**

**Table 366: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Twist at free end (radians)</td>
<td>0.0751</td>
<td>0.0751</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Twist in a Tapered Tube.STD is typically installed with the program.

**STAAD SPACE TORSION ON CONICAL SHAFT**

**START JOB INFORMATION**

**ENGINEER DATE 18-Sep-18**

**END JOB INFORMATION**

* * * REVIEW PROBLEM 3.120, PAGE 149. TWIST AT FREE END SHOULD BE ABOUT 0.0751 RADIAN

**UNIT METER KN**

**JOINT COORDINATES**

1 0 0 0; 2 0 2 0;

**MEMBER INCIDENCES**

1 1 2;

**UNIT MMS KN**

**MEMBER PROPERTY AMERICAN**

1 PRIS ROUND STA 80 END 40 THI 10

**UNIT METER KN**

**DEFINE MATERIAL START**

**ISOTROPIC MATERIAL1**

E 2e+08

POISSON 0.3

**END DEFINE MATERIAL**

**UNIT MMS KN**

**CONSTANTS**

**MATERIAL MATERIAL1 ALL SUPPORTS**

1 FIXED
V. Forces on a Propped Cantilever 1

To find the deflection and member forces due to an applied load on a propped cantilever beam with a compression only support.

Reference

Hand calculation using the following reference:


Problem

A cantilever beam with an end support capable of resisting only a compressive force is analyzed for two concentrated loads at 0.6xL:

1. +0.5 lbs (up), and
2. -0.5 lbs (down)

\[
E = 10(10)^6 \\
\text{width} = 0.6 \text{ in} \\
\text{depth} = 0.3 \text{ in} \\
L = 4 \text{ ft}
\]

*Note: A dummy member is used to represent a compression only support*
Theoretical Solution

Load Case 1

General solution equations found on p.2-121 of the reference.

\[ M(x < b) = -P(b - x) = -0.5lb(28.8 - x) \]

At the rigid support, \( M = 14.4 \text{ in} \cdot \text{lb} \).

By inspection, fixed end shear is equal to load value = 0.50 lb.

\[
\Delta(x < b) = \frac{Px^2}{6EI} \left( 3x - b \right) = \frac{0.5lb(28.8\text{ in})^2}{6\left(10,000,000\text{ psi}\right)(0.00135\text{ in}.)^4} \left[ 3\left(28.8\text{ in.}\right) - 28.8\text{ in.} \right] = 0.295\text{ in.
}

Load Case 2

General solution equations found on p.2-118 of the reference

\[
R_1 = \frac{Pb}{2\left(48\text{ in.}\right)^3} \left( a + 2\ell \right) = \frac{0.5lb(48\text{ in.})^2}{2\left(48\text{ in.}\right)^3} \left[ 19.2\text{ in.} + 2\left(48\text{ in.}\right) \right] = 0.216\text{ lb.
}
\]

\[ R_2 = P \cdot R_1 = 0.50\text{ lb} \cdot 0.216\text{ lb} = 0.284\text{ lb.
}

Moment at rigid support:

\[
M(x < b) = R_1(\ell - x) - P(\ell - x - a) = 0.216lb(48\text{ in.} - 0) - 0.5lb(48\text{ in.} - 0 - 19.2\text{ in.}) = -4.032\text{ in} \cdot \text{lb
}
\]

Deflection at point of load:

\[
\Delta(x < b) = \frac{-Pax^2}{12EI\ell^3} \left( 3t^3 - 3a^2\ell + a^2x \right) = \frac{-0.5lb(19.2\text{ in.})(28.8\text{ in.})^3}{12\left(10,000,000\text{ psi}\right)(0.00135\text{ in.})^4} \left[ 3(48\text{ in.})^3 - 3(48\text{ in.})(28.8\text{ in.}) - 3(19.2\text{ in.})(48\text{ in.}) + (19.2\text{ in.})^2(28.8\text{ in.}) \right] = -0.040\text{ in.
}
\]
Comparison

Table 367: Comparison of results

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>LC 1</td>
<td>Moment at fixed end (in·lb)</td>
<td>-14.4</td>
<td>-14.4</td>
<td>none</td>
</tr>
<tr>
<td>LC 1</td>
<td>Shear at fixed end (lb)</td>
<td>0.50</td>
<td>0.50</td>
<td>none</td>
</tr>
<tr>
<td>LC 1</td>
<td>Deflection at load (in)</td>
<td>0.295</td>
<td>0.295</td>
<td>none</td>
</tr>
<tr>
<td>LC 2</td>
<td>Moment at fixed end (in·lb)</td>
<td>4.032</td>
<td>4.03</td>
<td>none</td>
</tr>
<tr>
<td>LC 2</td>
<td>Shear at fixed end (lb)</td>
<td>0.284</td>
<td>0.28</td>
<td>1.4%</td>
</tr>
<tr>
<td>LC 2</td>
<td>Deflection at load (in)</td>
<td>-0.040</td>
<td>-.0401</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Forces on a Propped Cantilever 1.STD is typically installed with the program.

**STAAD PLANET :A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT**

**START JOB INFORMATION**

**ENGINEER** DATE 18-Sep-18

**END JOB INFORMATION**

***********************************************
* The end support is to be defined as a     *
* compression only support. A dummy member,*
* #7, is set as compression only to model   *
* this.                                     *
*********************************************

**SET NL 2**

**UNIT INCHES POUND**

**JOINT COORDINATES**

1 0 0 0; 2 9.6 0 0; 3 19.2 0 0; 4 28.8 0 0; 5 38.4 0 0; 6 48 0 0;
7 48 -4 0;

**MEMBER INCIDENCES**

1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7;

**MEMBER PROPERTY AMERICAN**

1 TO 5 PRIS YD 0.3 ZD 0.6

6 PRIS YD 40

**MEMBER COMPRESSION**

6
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.33
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
7 FIXED
*7 PINNED
LOAD 1 UPWARD
JOINT LOAD
4 FY 0.5
PERFORM ANALYSIS
PRINT MEMBER FORCES LIST 1 TO 5
PRINT JOINT DISPLACEMENTS LIST 1 TO 6
PRINT SUPPORT REACTION
CHANGE
LOAD 2 DOWNWARD
JOINT LOAD
4 FY -0.5
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES LIST 1 TO 5
PRINT JOINT DISPLACEMENTS LIST 1 TO 6
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>-0.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-14.40</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>0.00</td>
<td>-0.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>9.60</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>0.00</td>
<td>-0.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>4.80</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>4</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>6</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT ***************

39. PRINT JOINT DISPLACEMENTS LIST 1 TO 6
JOINT DISPLACE LIST
1
:A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00000</td>
<td>0.04370</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00853</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00000</td>
<td>0.15293</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.01365</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00000</td>
<td>0.29494</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.01536</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00000</td>
<td>0.44239</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.01536</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.00000</td>
<td>0.58985</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.01536</td>
</tr>
</tbody>
</table>

Verification Examples
V.01 Beams

STAAD.Pro
3207
User Manual
### SUPPORT REACTIONS - UNIT POUND INCH

**STRUCTURE TYPE = PLANE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>-0.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-14.40</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**NOTE - Tension/Compression converged after 1 iterations, Case = 2**

### MEMBER END FORCES

**STRUCTURE TYPE = PLANE**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>0.28</td>
<td>0.00</td>
<td>0.00</td>
<td>4.03</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>2</td>
<td>0.00</td>
<td>-0.28</td>
<td>0.00</td>
<td>0.00</td>
<td>-1.31</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>2</td>
<td>0.00</td>
<td>-0.28</td>
<td>0.00</td>
<td>0.00</td>
<td>1.33</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>3</td>
<td>0.00</td>
<td>-0.28</td>
<td>0.00</td>
<td>0.00</td>
<td>-1.42</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>4</td>
<td>0.00</td>
<td>-0.22</td>
<td>0.00</td>
<td>0.00</td>
<td>4.15</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>5</td>
<td>0.00</td>
<td>-0.22</td>
<td>0.00</td>
<td>0.00</td>
<td>-1.42</td>
<td></td>
</tr>
</tbody>
</table>

### JOINT DISPLACEMENTS - INCH RADIANS

**STRUCTURE TYPE = PLANE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.00000</td>
<td>-0.01066</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.00190</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>0.00000</td>
<td>-0.03024</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.00186</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>0.00000</td>
<td>-0.04012</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00012</td>
</tr>
</tbody>
</table>
V. Torsion on a Stepped Cantilever

To find end rotation due to torques on a stepped cantilever shaft.

Reference
Hand calculation using the following reference:

Problem
A stepped shaft is subjected to torques as shown in the figure. The material has shear modulus of elasticity $G = 80$ Gpa. Determine the angle of twist $\theta_x$ in degrees at the free end.

![Figure 351: Cantilevered member subject to torsional loads](image)

Theoretical Solution
Moment of Inertia:

\[
I_{p1} = \frac{n(80\text{mm}/2)^4}{2} = 4.021(10)^6\text{mm}^4
\]

\[
I_{p2} = \frac{n(60\text{mm}/2)^4}{2} = 1.272(10)^6\text{mm}^4
\]

\[
I_{p2} = \frac{n(40\text{mm}/2)^4}{2} = 0.251(10)^6\text{mm}^4
\]

Angle of twist is given by:
\[ \theta = \sum_{i} \frac{L_i T_i}{G I_p} \]

\[ = \frac{5 \times 800 \text{N-mm}(500 \text{mm})}{4.021 \times (10^6 \text{mm}^4 \text{GPa})} + \frac{2 \times 800 \text{N-mm}(500 \text{mm})}{1.272 \times (10^6 \text{mm}^4 \text{GPa})} + \frac{2 \times 800 \text{N-mm}(500 \text{mm})}{0.251 \times (10^6 \text{mm}^4 \text{GPa})} \]

\[ = 0.0090 + 0.0138 + 0.0199 = 0.0427 \]

which is equal to 2.446°.

**Comparison**

Table 368: Comparison of results

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of twist (rad.)</td>
<td>0.0427</td>
<td>0.0427</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file \texttt{C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Torsion on a Stepped Cantilever.STD} is typically installed with the program.

```
* REFERENCE: MECHANICS OF MATERIALS, GERE AND TIMOSHENKO, 2ND EDITION
*            PROBLEM 3.3-1 PAGE 171
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0.5 0 0; 3 1 0 0; 4 1.5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
UNIT MMS KN
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPE OD 80 ID 0
2 TABLE ST PIPE OD 60 ID 0
3 TABLE ST PIPE OD 40 ID 0
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 200
POISSON 0.25
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
UNIT MMS KN
LOAD 1 TORSIONAL MOMENT
JOINT LOAD
```
V. Forces on a Propped Cantilever 2

To find deflections, stress and support reactions due to a uniform load on a beam with one end fixed and the other end supported by a roller.

Reference

Hand calculation using the following reference:


Problem

A horizontal beam of length = 100 in, area = 4 in$^2$, height = 2 in, and moment of inertia = 1.3333 in$^4$ is simply supported at one end and fixed at the other end. The beam is subjected to a uniform loading. Determine the deflection $\delta$ at $x = 42.15$ in., the slope $\theta$ at end A, the maximum bending stress $\sigma_{bend}$ in the beam and the support reactions.

\[
E = 30\times(10)^6 \text{ psi}
\]

\[
\text{Density} = 0.2821 \text{ lbs/in}^3
\]
Comparison

Table 369: Comparison of results

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reaction at Node 1 (lb)</td>
<td>42.31</td>
<td>42.31</td>
<td>none</td>
</tr>
<tr>
<td>Reaction at Node 3 (lb)</td>
<td>70.52</td>
<td>70.52</td>
<td>none</td>
</tr>
<tr>
<td>Moment at Node 3 (in-lb)</td>
<td>1,410.4</td>
<td>1410.4</td>
<td>none</td>
</tr>
<tr>
<td>Bending Stress, $\sigma_{bend}$ at Node 2 (psi)</td>
<td>585.9</td>
<td>585.9</td>
<td>none</td>
</tr>
<tr>
<td>Rotation at Node 1 (rad.)</td>
<td>-.000588</td>
<td>-.00059</td>
<td>none</td>
</tr>
<tr>
<td>Deflection at Node 2 (in.)</td>
<td>-.01528</td>
<td>-.01528</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Forces on a Propped Cantilever 2.STD is typically installed with the program.

STAAD PLANE : A FIXED-ROLLER BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: ROARK AND YOUNG, PAGE 109, NO. 23.
*
INPUT WIDTH 72
UNIT INCHES POUND

JOINT COORDINATES
1 0 0 0; 2 42.15 0 0; 3 100 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 4 IZ 1.3333 YD 2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
DENSITY 0.282072
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
3 FIXED
1 PINNED
LOAD 1 SELF WEIGHT
SELFWEIGHT Y -1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PRINT MEMBER STRESSES ALL
FINISH

--- STAAD Output ---

<table>
<thead>
<tr>
<th>JOINT DISPLACEMENT (INCH RADIANS)</th>
<th>STRUCTURE TYPE = PLANE</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT</td>
<td>LOAD</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
</tr>
</tbody>
</table>

:A FIXED-ROLLER BEAM -- PAGE NO.

SUPPORT REACTIONS -UNIT POUN INCH | STRUCTURE TYPE = PLANE |
-----------------------------------|------------------------|
<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>70.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-1410.36</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>42.31</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

:A FIXED-ROLLER BEAM -- PAGE NO.

MEMBER END FORCES | STRUCTURE TYPE = PLANE |
--------------------|------------------------|
<p>| ALL UNITS ARE -- | POUN INCH (LOCAL) |</p>
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>42.31</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>781.13</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>0.00</td>
<td>-5.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-781.13</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>70.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-1410.36</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************
V. Axially Loaded Column

To find support reactions due to an axial load applied at two locations on a column with both ends pinned.

Reference

Hand calculation using the following reference:


Problem

Find the support reactions at the end joints 1 and 4.

$$ E = 30 \times (10)^6 \text{ psi} $$
Verification Examples

V.01 Beams

Figure 353: Column A) problem sketch and B) mathematical model

Comparison

Table 370: Comparison of results

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reaction at Node 1 (lb)</td>
<td>600</td>
<td>600</td>
<td>none</td>
</tr>
<tr>
<td>Reaction at Node 4 (lb)</td>
<td>900</td>
<td>900</td>
<td>none</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Axially Loaded Column.STD is typically installed with the program.

STAAD PLANE :A PINNED-PINNED COLUMN
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE: Timoshenko, Strength of Materials, Part 1
* Problem 10, Page 26
* INPUT WIDTH 79
UNIT INCHES POUND
JOIN COORDINATES
1 48 72 0; 2 48 76 0; 3 48 79 0; 4 48 82 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 1 IZ 0.0001
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 PINNED
LOAD 1 AXIAL LOAD
JOIN LOAD
3 FY -1000
2 FY -500
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>SUPPORT REACTIONS -UNIT POUN INCH</th>
<th>STRUCTURE TYPE = PLANE</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD</td>
<td>FORCE-X</td>
</tr>
<tr>
<td>1 1</td>
<td>0.00</td>
</tr>
<tr>
<td>4 1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

V. Tee Shaped Cantilever

To find the stress due to an applied moment at the free end of a cantilever beam with inverted tee section.
Reference

Hand calculation using the following reference:

Problem

Find the maximum bending stress in the beam.

\[ E = 30 \times (10)^6 \text{ psi} \]
\[ b = 1.5 \text{ in.}, h = 8 \text{ in.}, L = 10 \text{ in.} \]
\[ M = 1,000,000 \text{ in. \cdot lb} \]

Comparison

Table 371: Comparison of results

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending stress, ( \sigma ) (psi)</td>
<td>700</td>
<td>700</td>
<td>none</td>
</tr>
</tbody>
</table>

Figure 354: Beam A) problem sketch and B) mathematical model
**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Tee Shaped Cantilever.STD is typically installed with the program.

STAAD PLANE : A CANTILEVERED BEAM OF INVERTED TEE SECTION

START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* * * REFERENCE: CRANDALL & DAHL, AN INTRODUCTION TO THE MECHANICS OF SOLIDS, PAGE 294, EX. 7.2 * * *

INPUT WIDTH 79
UNIT INCHES POUND

JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 PRIS YD 20 ZD 9 YB 16 ZB 1.5
DEFINE MATERIAL START
ISOTROPIC MATERIAL 1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL

CONSTANTS
BETA 180 ALL
MATERIAL MATERIAL 1 ALL

SUPPORTS
1 FIXED
LOAD 1 CONSTANT MOMENT
JOINT LOAD
2 MZ 100000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT MEMBER STRESSES ALL
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>MEMB</th>
<th>LD</th>
<th>SECT</th>
<th>AXIAL</th>
<th>BEND-Y</th>
<th>BEND-Z</th>
<th>COMBINED</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>700.0</td>
<td>700.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>0</td>
<td>0.0 C</td>
<td>0.0</td>
<td>700.0</td>
<td>700.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**V. Beam on Elastic Foundation**

To find deflection and stress at the center due to a uniform, static load on a simply supported beam on elastic foundation.
Reference

Hand calculation using the following reference:


Problem

Find the vertical deflection and bending stress at the center of the beam. Spacing between

\[ E = 30 \times (10)^6 \text{ psi} \]
\[ b = 1.0 \text{ in.}, h = 7.114 \text{ in.}, L = 240 \text{ in.} \]
\[ w_u = 43.3 \text{ lb/in.} \]

![Figure 355: One-half beam for mathematical model](image)

Comparison

Table 372: Comparison of results

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending stress, ( \sigma ) (psi)</td>
<td>18,052</td>
<td>18,053.29</td>
<td>none</td>
</tr>
<tr>
<td>Vertical deflection (in.)</td>
<td>1.0453</td>
<td>1.04549</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Beam on Elastic Foundation.STD is typically installed with the program.

```
STAAD SPACE :A SIMPLY SUPPORTED BEAM ON ELASTIC FOUNDATION
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
```
*REFERENCE: PETERSON, EASE2, EXAMPLE PROBLEM MANUAL*

INPUT WIDTH 72
UNIT INCHES POUND

JOINT COORDINATES
1 0 0 0; 2 0 7.114 0; 3 0 0 6; 4 0 7.114 6; 5 0 0 12; 6 0 7.114 12;
7 0 0 18; 8 0 7.114 18; 9 0 0 24; 10 0 7.114 24; 11 0 0 30;
12 0 7.114 30; 13 0 0 36; 14 0 7.114 36; 15 0 0 42; 16 0 7.114 42;
17 0 0 48; 18 0 7.114 48; 19 0 0 54; 20 0 7.114 54; 21 0 0 60;
22 0 7.114 60; 23 0 0 66; 24 0 7.114 66; 25 0 0 72; 26 0 7.114 72;
27 0 0 78; 28 0 7.114 78; 29 0 0 84; 30 0 7.114 84; 31 0 0 90;
32 0 7.114 90; 33 0 0 96; 34 0 7.114 96; 35 0 0 102; 36 0 7.114 102;
37 0 0 108; 38 0 7.114 108; 39 0 0 114; 40 0 7.114 114; 41 0 0 120;
42 0 7.114 120;

ELEMENT INCIDENCES SHELL

1 1 2 4 3; 2 3 4 6 5; 3 5 6 8 7; 4 7 8 10 9; 5 9 10 12 11;
6 11 12 14 13; 7 13 14 16 15; 8 15 16 18 17; 9 17 18 20 19;
10 19 20 22 21; 11 21 22 24 23; 12 23 24 26 25; 13 25 26 28 27;
14 27 28 30 29; 15 29 30 32 31; 16 31 32 34 33; 17 33 34 36 35;
18 35 36 38 37; 19 37 38 40 39; 20 39 40 42 41;

ELEMENT PROPERTY

1 TO 20 THICKNESS 1

DEFINE MATERIAL START

ISOTROPIC MATERIAL1

E 3e+07

POISSON 0.290909

END DEFINE MATERIAL

CONSTANTS

MATERIAL MATERIAL1 ALL

SUPPORTS

1 FIXED BUT KFY 78.125

41 FIXED BUT FZ MX

2 FIXED BUT FY

3 5 7 9 11 13 15 17 19 21 23 25 27 29 31 33 35 37 -

39 FIXED BUT FZ MX KFY 156.25

LOAD 1 UNIFORM LOAD OF 43.4 LBS/IN

JOINT LOAD

4 6 8 10 12 14 16 18 20 22 24 26 28 30 32 34 36 38 40 FY 260.4

2 42 FY 130.2

PERFORM ANALYSIS

PRINT JOINT DISPLACEMENTS LIST 1 2

PRINT ELEMENT JOINT STRESSES LIST 1 2

FINISH

STAAD Output

JOINT DISPLACEMENT (INCH RADIANS) STRUCTURE TYPE = SPACE

------------------

JOINT LOAD X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN

1 1 0.0000 1.04548 0.0000 0.0000 0.0000 0.0000

2 1 0.0000 1.04549 0.0000 0.0000 0.0000 0.0000

*************** END OF LATEST ANALYSIS RESULT ***************

47. PRINT ELEMENT JOINT STRESSES LIST 1 2

ELEMENT JOINT STRESSES LIST

:A SIMPLY SUPPORTED BEAM ON ELASTIC FOUNDATION -- PAGE NO.
### Verification Examples

**V.01 Beams**

#### ELEMENT STRESSES

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQUARE</th>
<th>SQUARE</th>
<th>VONMISES</th>
<th>TRESCA</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>37.23</td>
<td>37.23</td>
<td>35.30</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>37.85</td>
<td>37.85</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX=36.57</td>
<td>SMIN=-1.27</td>
<td>TMAX=18.92</td>
<td>ANGLE=-10.6</td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=36.57</td>
<td>SMIN=-1.27</td>
<td>TMAX=18.92</td>
<td>ANGLE=-10.6</td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES

<table>
<thead>
<tr>
<th>MAXIMUM</th>
<th>MINIMUM</th>
<th>MAXIMUM</th>
<th>MAXIMUM</th>
<th>MAXIMUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRINCIPAL</td>
<td>PRINCIPAL</td>
<td>SHEAR</td>
<td>VONMISSE</td>
<td>TRESCA</td>
</tr>
<tr>
<td>STRESS</td>
<td>STRESS</td>
<td>STRESS</td>
<td>STRESS</td>
<td>STRESS</td>
</tr>
<tr>
<td>1.805341E+04</td>
<td>-1.805341E+04</td>
<td>9.044362E+03</td>
<td>5.004878E+01</td>
<td>5.409544E+01</td>
</tr>
</tbody>
</table>
V. Stresses in a Circular Beam

Find deflections and stress at the center of a locomotive axle.

Reference
p. 94, problems 1, 2.

Problem
Determine the maximum stress in a locomotive axle (as shown in the figure) as well as the deflection at the middle of the axle.

- Diameter = 10 in.
- \( P = 26,000 \text{ lbf} \)
- \( E = 30 \times (10)^6 \text{ psi} \)
- \( L_1 = 13.5 \text{ in.}, L_2 = 59 \text{ in.} \)

![Figure 356: Locomotive axle model](image)

Comparison

<table>
<thead>
<tr>
<th>Table 373: Comparison of results</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Result Type</strong></td>
</tr>
<tr>
<td>( \sigma ) (psi), Node 3</td>
</tr>
<tr>
<td>( \delta ) (in.), Node 3</td>
</tr>
</tbody>
</table>

* The value is recalculated.
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Stresses in a Circular Beam.STD is typically installed with the program.

STAAD PLANE : STRESSES IN A CIRCULAR BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
*  REFERENCE 'STRENGTH OF MATERIALS' PART-1 BY S. TIMOSHENKO
*  PAGE 97 PROBLEM NO. 1 AND 2. ANSWERS ARE 3580 FOR MAX. STRESS
*  AND 0.104 INCH FOR MAX. DEFLECTION.
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 13.5 0 0; 3 43 0 0; 4 72.5 0 0; 5 86 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5;
MEMBER PROPERTY AMERICAN
1 TO 4 TABLE ST PIPE OD 10 ID 0
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
2 4 PINNED
LOAD 1
JOINT LOAD
1 5 FY -26000
PERFORM ANALYSIS
PRINT MEMBER STRESSES
PRINT JOINT DISPLACEMENTS
FINISH

STAAD Output

MEMBER STRESSES
--------------
ALL UNITS ARE POUN/SQ INCH

Memb  LD  Sect  Axial  Bend-Y  Bend-Z  Combined  Shear-Y  Shear-Z
1 1  .0  0.0  0.0  0.0  0.0  441.4  0.0
   1.0  0.0 C  0.0  3575.3  3575.3  441.4  0.0
2 1  .0  0.0  0.0  0.0  3575.3  3575.3  441.4  0.0
   1.0  0.0 C  0.0  3575.3  3575.3  441.4  0.0
3 1  .0  0.0  0.0  0.0  3575.3  3575.3  441.4  0.0
   1.0  0.0 C  0.0  3575.3  3575.3  441.4  0.0
4 1  .0  0.0  0.0  0.0  3575.3  3575.3  441.4  0.0
   1.0  0.0 C  0.0  3575.3  3575.3  441.4  0.0
****** END OF LATEST ANALYSIS RESULT *******
V. End Moments in a Non Uniform Beam

To find end moments due to a uniform load on a beam with nonuniform sections, fixed at both ends.

Reference

Hand calculation using the following reference:

Problem

Find the moment at the supports. Assume for input a unit width for the beam. Depths are as shown.

\[ E = 30 \times (10)^6 \text{ psi} \]
\[ w = 4 \text{ k/ft} \]
\[ d_1 = 10 \text{ in.}, d_2 = 20 \text{ in.} \]
\[ L_1 = 12 \text{ ft}, L_2 = 8 \text{ ft} \]
Comparison

Table 374: Comparison of results

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment at Node 1 (in·lb)</td>
<td>-98.2</td>
<td>-96.93</td>
<td>1.3%</td>
</tr>
<tr>
<td>Moment at Node 2 (in·lb)</td>
<td>-217.2</td>
<td>-220.42</td>
<td>1.5%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\End Moments in a Non Uniform Beam.STD is typically installed with the program.

STAAD PLANE :A FIXED-FIXED BEAM OF UNIFORM AND TAPERED SECTIONS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE: MCCORMAC, J.C., STRUCTURAL ANALYSIS, INTEXT
* EDUCATIONAL PUBLISHERS, NEW YORK, 3RD EDITION, 1975
V. Stresses in a Tapered Cantilever

To find the maximum deflection and principal stress due to a load on the free end of a cantilever beam with tapered section.

Reference

Hand calculation using the following reference:


Problem

Find the maximum deflection, $\delta$, and the principal normal stress, $\sigma$, in the beam.
E = 30 \times (10)^6 \text{ psi}

P = 10 \text{ lb}

d = 3 \text{ in.}, b = 0.5 \text{ in.}

L = 20

Figure 358: Beam with a tapering cross section

Comparison

Table 375: Comparison of results

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum deflection at free end (in.)</td>
<td>-0.04267</td>
<td>-0.04265</td>
<td>none</td>
</tr>
<tr>
<td>Principle stress (psi)</td>
<td>1,600</td>
<td>1,600</td>
<td>none</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Stresses in a Tapered Cantilever.STD is typically installed with the program.

STAAD PLANE : A CANTILEVER BEAM OF TAPERED SECTION

**START JOB INFORMATION**

ENGINEER DATE 18-Sep-18

**END JOB INFORMATION**

* * * REFERENCE: HARRIS, C.O., INTRODUCTION TO STRESS ANALYSIS,
* THE MACMILLAN CO., NEW YORK, 1956

* * *

* USING A TAPERED BEAM ELEMENT
* *

**INPUT WIDTH 72**

UNIT INCHES POUND

**JOINT COORDINATES**

1 0 0 0; 2 2 0 0; 3 4 0 0; 4 6 0 0; 5 8 0 0; 6 10 0 0; 7 12 0 0;
8 14 0 0; 9 16 0 0; 10 18 0 0; 11 20 0 0;

**MEMBER INCIDENCES**

1 11 10; 2 10 9; 3 9 8; 4 8 7; 5 7 6; 6 6 5; 7 5 4; 8 4 3; 9 3 2;
10 2 1;

**MEMBER PROPERTY AMERICAN**

1 TAPERED 3 0.5 2.7 0.5 0.01
2 TAPERED 2.7 0.5 2.4 0.5 0.01
3 TAPERED 2.4 0.5 2.1 0.5 0.01
4 TAPERED 2.1 0.5 1.8 0.5 0.01
5 TAPERED 1.8 0.5 1.5 0.5 0.01
6 TAPERED 1.5 0.5 1.2 0.5 0.01
7 TAPERED 1.2 0.5 0.9 0.5 0.01
8 TAPERED 0.9 0.5 0.6 0.5 0.01
9 TAPERED 0.6 0.5 0.3 0.5 0.01
10 TAPERED 0.3 0.5 0.03 0.5 0.01

**DEFINE MATERIAL START**

ISOTROPIC MATERIAL1

E 3e+07

POISSON 0.290909

**END DEFINE MATERIAL**

**CONSTANTS**

BETA 90 ALL

**MATERIAL MATERIAL1 ALL**

**SUPPORTS**

11 FIXED

**LOAD 1 POINT LOAD AT TIP**

JOINT LOAD

1 FY -10

**PERFORM ANALYSIS**

PRINT MEMBER PROPERTIES ALL

PRINT MEMBER FORCES

PRINT SUPPORT REACTION

SECTION 0.5 ALL

PRINT MEMBER SECTION FORCES ALL

PRINT JOINT DISPLACEMENTS LIST 1
PRINT MEMBER STRESSES ALL
FINISH

STAAD Output

MEMBER PROPERTIES. UNIT - INCH
-----------------
MEMB PROFILE AX/ IZ/ IY/ IX/
     AY     AZ     SZ     SY
1   TAP ERED  1.42   0.97   0.03   0.12
     1.42   0.01   0.68   0.12
2   TAP ERED  1.28   0.70   0.03   0.11
     1.28   0.01   0.54   0.11
3   TAP ERED  1.12   0.48   0.02   0.09
     1.12   0.01   0.42   0.09
4   TAP ERED  0.97   0.31   0.02   0.08
     0.97   0.01   0.32   0.08
5   TAP ERED  0.82   0.19   0.02   0.07
     0.82   0.01   0.23   0.07
6   TAP ERED  0.68   0.11   0.01   0.06
   0.68   0.01   0.15   0.06
7   TAP ERED  0.52   0.05   0.01   0.04
   0.52   0.01   0.09   0.04
8   TAP ERED  0.38   0.02   0.01   0.03
   0.38   0.01   0.05   0.03
9   TAP ERED  0.23   0.01   0.00   0.02
   0.23   0.01   0.02   0.02
10  TAP ERED  0.08   0.00   0.00   0.01
   0.08   0.01   0.00   0.01
*********** END OF DATA FROM INTERNAL STORAGE ***********

45. PRINT MEMBER FORCES

MEMBER FORCES
:A CANTILEVER BEAM OF TAPERED SECTION
-----------------
MEMBER END FORCES STRUCTURE TYPE = PLANE
-----------------
ALL UNITS ARE -- POUN INCH (LOCAL )
MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
1  1    11  0.00  0.00  -10.00  0.00  200.00  0.00
    10  0.00  0.00  10.00  0.00  -180.00  0.00
2  1    10  0.00  0.00  -10.00  0.00   180.00  0.00
    9  0.00  0.00  10.00  0.00   -160.00  0.00
3  1     9  0.00  0.00  -10.00  0.00   160.00  0.00
    8  0.00  0.00  10.00  0.00   -140.00  0.00
4  1     8  0.00  0.00  -10.00  0.00   140.00  0.00
    7  0.00  0.00  10.00  0.00   -120.00  0.00
5  1     7  0.00  0.00  -10.00  0.00   120.00  0.00
    6  0.00  0.00  10.00  0.00   -100.00  0.00
6  1     6  0.00  0.00  -10.00  0.00   100.00  0.00
    5  0.00  0.00  10.00  0.00   -80.00  0.00
7  1     5  0.00  0.00  -10.00  0.00    80.00  0.00
    4  0.00  0.00  10.00  0.00   -60.00  0.00
8  1     4  0.00  0.00  -10.00  0.00    60.00  0.00
    3  0.00  0.00  10.00  0.00   -40.00  0.00
9  1     3  0.00  0.00  -10.00  0.00    40.00  0.00
    2  0.00  0.00  10.00  0.00   -20.00  0.00

Verification Examples
V.01 Beams

STAAD.Pro 3229 User Manual
**Verification Examples**

**V.01 Beams**

```
10 1 2 0.00 0.00 -10.00 0.00 20.00 0.00
1 0.00 0.00 10.00 0.00 -0.00 0.00

*************** END OF LATEST ANALYSIS RESULT ***************

46. PRINT SUPPORT REACTION

SUPPORT REACTION
:A CANTILEVER BEAM OF TAPERED SECTION  -- PAGE NO.

SUPPORT REACTIONS -UNIT POUN INCH  STRUCTURE TYPE = PLANE
-----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z   MOM-X   MOM-Y   MOM Z
11  1     0.00     10.00      0.00      0.00      0.00   -200.00

*************** END OF LATEST ANALYSIS RESULT ***************

47. SECTION 0.5 ALL

48. PRINT MEMBER SECTION FORCES ALL

MEMBER FORCES AT INTERMEDIATE SECTIONS
--------------------------------------
ALL UNITS ARE -- POUN INCH
MEMB  LOAD  SEC       AXIAL   SHEAR-Y   SHEAR-Z       MOM-X       MOM-Y       MOM-Z
1    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
190.00        0.00
2    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
170.00        0.00
3    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
150.00        0.00
4    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
130.00        0.00
5    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
110.00        0.00
6    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
 90.00        0.00
7    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
 70.00        0.00
8    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
 50.00        0.00
9    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
 30.00        0.00
10    1  0.50       0.00      0.00    -10.00        0.00        0.00        0.00
 10.00        0.00

*************** END OF LATEST ANALYSIS RESULT ***************

49. PRINT JOINT DISPLACEMENTS LIST 1

JOINT  DISPLACE LIST  1
:A CANTILEVER BEAM OF TAPERED SECTION  -- PAGE NO.

7  
JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = PLANE
-----------------
JOINT LOAD X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN
1  1    0.00000   -0.04265   0.00000    0.00000   0.00000    0.0419

*************** END OF LATEST ANALYSIS RESULT ***************

50. PRINT MEMBER STRESSES ALL

MEMBER STRESSES ALL
:A CANTILEVER BEAM OF TAPERED SECTION  -- PAGE NO.

8
MEMBER STRESSES
-----------------
```
### V. Stresses in a Cable due to Thermal Loading

A rigid bar is suspended by two copper wires and one steel wire. Find the stresses in the wires due to a rise in temperature.

#### Reference


#### Problem

Assuming the horizontal member to be very rigid, determine the stresses in the copper and steel wires if the temperature rise is 10° F. Members 1 and 3 are copper and member 2 is steel.

\[ E_{\text{steel}} = 30 \times (10)^6 \text{ psi, } E_{\text{copper}} = 16 \times (10)^6 \text{ psi} \]
\[ \alpha_{\text{steel}} = 70 \times 10^{-7} \text{ in/in/°F}, \quad \alpha_{\text{copper}} = 92 \times 10^{-7} \text{ in/in/°F} \]

\[ A_X = 0.1 \text{ in}^2 \]

\[ w = 400 \text{ lbf/in.} \]

\[ L = 20 \text{ in.} \]

\[ d = 5 \text{ in.} \]

Tip: When modeling, assume a large moment of inertia for the horizontal rigid member and distribute of the concentrated load as uniform.

\[
\begin{align*}
A_X &= 0.1 \text{ in}^2 \\
w &= 400 \text{ lbf/in.} \\
L &= 20 \text{ in.} \\
d &= 5 \text{ in.}
\end{align*}
\]

Comparison

Table 376: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_{\text{Steel}} ) (psi)</td>
<td>19,695</td>
<td>19,698.3</td>
<td>negligible</td>
</tr>
<tr>
<td>( \sigma_{\text{Copper}} ) (psi)</td>
<td>10,152</td>
<td>10,150.8</td>
<td>negligible</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Stresses in a Cable due to Thermal Loading.STD is typically installed with the program.

STAAD PLANE: A RIGID BAR SUSPENDED BY TWO COPPER WIRES AND A STEEL WIRE
START JOB INFORMATION
ENGINEER DATE 09-Oct-17
END JOB INFORMATION
*
* REFERENCE: 'STRENGTH OF MATERIALS', TIMOSHENKO (PART 1), PAGE 30, PROB 9.
THE ANSWERS ARE 19700 PSI AND 10200 PSI.

UNIT INCHES POUND

JOINT COORDINATES
1 0 20 0; 2 5 20 0; 3 10 20 0; 4 0 0 0; 5 5 0 0; 6 10 0 0;

MEMBER INCIDENCES
1 1 4; 2 2 5; 3 3 6; 4 4 5; 5 5 6;

MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 0.1 IZ 0.0001
4 5 PRIS AX 1 IZ 100

DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1.6e+007
POISSON 0.230769
ALPHA 9.2e-006

ISOTROPIC MATERIAL2
E 3e+007
POISSON 0.290909
ALPHA 7e-006

END DEFINE MATERIAL

CONSTANTS
MATERIAL MATERIAL1 MEMB 1 3
MATERIAL MATERIAL2 MEMB 2 4 5

SUPPORTS
1 2 3 Fixed

MEMBER RELEASE
1 TO 3 START MPZ 0.99
1 TO 3 END MZ

LOAD 1 VERT LOAD + TEMP LOAD
MEMBER LOAD
4 5 UNI Y -400
TEMPERATURE LOAD
1 TO 3 TEMP 10

PERFORM ANALYSIS
PRINT MEMBER STRESSES LIST 1 TO 3
FINISH

STAAD Output

MEMBER STRESSES

---------------
ALL UNITS ARE POUN/SQ INCH

MEMB LD SECT AXIAL BEND-Y BEND-Z COMBINED SHEAR-Y SHEAR-Z
1 1 .0 10150.8 T 0.0 0.0 10150.8 0.0 0.0
1.0 10150.8 T 0.0 0.0 10150.8 0.0 0.0
2 1 .0 19698.3 T 0.0 0.0 19698.3 0.0 0.0
1.0 19698.3 T 0.0 0.0 19698.3 0.0 0.0
3 1 .0 10150.8 T 0.0 0.0 10150.8 0.0 0.0
1.0 10150.8 T 0.0 0.0 10150.8 0.0 0.0
V. Curved Beam

To find the out-of-plane deflection and stress in a Circular cantilever member with a concentrated load at the free end.

Reference

Hand calculation using the following reference:


Problem

Calculate the displacement at the free end and the bending stress at the fixed end due to a concentrated load producing out-of-plane bending.

\[
E = 30 \times (10)^6 \text{ psi}
\]

\[
P = 50 \text{ lb}
\]

\[
r = 100 \text{ in.}
\]
### Comparison

**Table 377: Comparison of results**

<table>
<thead>
<tr>
<th>Result</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum deflection at free end (in.)</td>
<td>2.648</td>
<td>2.64658</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Principle stress (psi)</td>
<td>6,366.0</td>
<td>6,638.1</td>
<td>4.3%</td>
<td></td>
</tr>
</tbody>
</table>

The result from a classical theory of a beam curved in plan is compared with the result from a piecewise linear set of beams, which closely resembles the behavior of a curved beam, but not exactly. Hence the difference in results. The difference may be further reduced to a certain level by discretizing the curved beam in more smaller subdivisions.

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Curved Beam.STD is typically installed with the program.

```
STAAD SPACE :A CURVED BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
*    REFERENCE: TIMOSHENKO, S., "STRENGTH OF MATERIALS, PART I, ELEMENTARY
*         THEORY AND PROBLEMS", 3RD EDITION, D. VAN NOSTRAND CO.,
*         INC., NEW YORK, 1955.
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 100 0 0; 2 99.619 0 -8.716; 3 98.481 0 -17.365; 4 96.593 0 -25.882;
5 93.969 0 -34.202; 6 90.631 0 -42.262; 7 86.604 0 -50.711;
8 81.915 0 -57.358; 9 76.604 0 -64.279; 10 70.711 0 -70.711;
11 64.279 0 -76.604; 12 57.358 0 -81.915; 13 50 0 -86.603;
```
V. Hanging Bar Axial Stress

Two vertical bars are supported by a rigid bar, which is pinned-supported on one end. Find stresses in vertical bars due to a load at the free end of the rigid bar.

Reference

Problem

TBars A and B are connected by rigid links to a fixed support at the top and to a rigid bar at the bottom. Determine the axial stresses in bars A and B when the load P is 177.92888 kN applied as shown.

\[
P = 177.93 \text{ kN}
\]

\[
A_A = 1,290.3 \text{ mm}^2
\]

\[
A_B = 1,612.9 \text{ mm}^2
\]

\[
E_A = 68.95 \text{ GPa}
\]

\[
E_B = 206.84 \text{ GPa}
\]

Assume the moment of inertia of member CD to be very large.

![Figure 361: Rigid bar hanging from a pair of rods](image)

Comparison

Table 378: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress in A (GPa)</td>
<td>0.17237</td>
<td>0.1723889</td>
<td>negligible</td>
</tr>
<tr>
<td>Stress in B (GPa)</td>
<td>0.20684</td>
<td>0.2068032</td>
<td>negligible</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Hanging Bar Axial Stress.STD is typically installed with the program.

STAAD PLANE :TWO VERTICAL MEMBERS SUPPORT A RIGID BAR
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MECHANICS OF MATERIALS, HIGDON, OHLEN, STILES, WEESE
* AND RILEY, 3RD EDITION, JOHN WILEY & SONS,
* PAGE 135, PROBLEM 3-37
*
UNIT METER NEWTON

JOINT COORDINATES
1 0 0 0; 2 0.127 0 0; 3 0.2032 0 0; 4 0.254 0 0; 5 0.127 0.254 0;
6 0.2032 0.254 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 0.032258 IZ 0.00208226
4 PRIS AX 0.00129032
5 PRIS AX 0.0016129
UNIT METER KN

DEFINE MATERIAL ISOTROPIC MATERIAL1
E 2.06843e+08
POISSON 0.290909

ISOTROPIC MATERIAL2
E 6.89477e+07
POISSON 0.329996
END DEFINE MATERIAL

UNIT METER NEWTON
CONSTANTS
MATERIAL MATERIAL1 MEMB 1 TO 3 5
MATERIAL MATERIAL2 MEMB 4
MEMBER TRUSS
4 5
SUPPORTS
4 PINNED
5 6 FIXED
LOAD 1 VERTICAL JOINT LOAD
JOINT LOAD
1 FY -177929
PERFORM ANALYSIS
UNIT METER KN
PRINT MEMBER STRESSES LIST 4 5
FINISH
V. Bent Cantilever Deflection

Find deflection due to load at the free end of a cantilever plane bent.

Reference


Problem

Find the vertical, horizontal and rotational deflection components of point A.

\[ E = 30,000 \text{ ksi} \]
\[ I = 200 \text{ in}^4 \]
\[ A = 10 \text{ in}^2 \]

*Figure 362: Bent plate frame*
### Comparison

#### Table 379: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection right, $\delta_x$ (in)</td>
<td>0.53</td>
<td>0.53056</td>
<td>none</td>
</tr>
<tr>
<td>Deflection down, $\delta_y$ (in)</td>
<td>1.16</td>
<td>-1.17109</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>Rotation, $\theta$ (rad)</td>
<td>0.0049</td>
<td>0.00488</td>
<td>none</td>
</tr>
</tbody>
</table>

#### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Bent Cantilever Deflection.STD is typically installed with the program.

```
STAAD PLANE : A CANTILEVER PLANE BENT
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE: INDETERMINATE STRUCTURAL ANALYSIS, KINNEY, 1957,
* ADISON-WESLEY PUBLISHING CO., PAGE 113, PROBLEM 4-38
*
UNIT FEET KIP
JOINT COORDINATES
  1  0  3  0; 2  0  0  0; 3  4  0  0; 4  14 10  0; 5  22 10  0;
MEMBER INCIDENCES
  1  1  2; 2  2  3; 3  4  3; 4  4  5;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
  1 TO 4 PRIS AX 10 IZ 200
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
  E  30000
POISSON  0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
  5 FIXED
LOAD 1 HORIZONTAL JOINT LOAD
JOINT LOAD
  1 FX 3
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1
FINISH
```
V. Bent Beam Thermal Loading

To find member forces and moments due to a temperature load on a Zee shaped plane bent.

Reference


Problem

Calculate reactions and maximum moments in the structure due to a temperature increase of 430 ºF. Do not consider shear deformation.

\[ E = 26,400 \text{ ksi} \]
\[ \alpha = 7.26744 \times 10^{-6} \text{ in/in}^\circ\text{F} \]
\[ \text{OD} = 12 \text{ in} \]
\[ \text{ID} = 10.255 \text{ in} \]
Comparison

Table 380: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal reaction (lbs)</td>
<td>8,980</td>
<td>8,949.43</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Vertical reaction (lbs)</td>
<td>7,756</td>
<td>7,729.42</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Moment at supports (in-lb)</td>
<td>783,750</td>
<td>781,127.75</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Moment at node 2 (in-lb)</td>
<td>1,077,656</td>
<td>1,073,932.12</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Bent Beam Thermal Loading.STD is typically installed with the program.

STAAD PLANE :A FIXED-FIXED ZEE SHAPED PLANE BENT BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
**Verification Examples**

**V.01 Beams**

```plaintext
END JOB INFORMATION
*
* REFERENCE: Seeley, F.B., and Smith, J.O., Advanced Mechanics of Materials,
*
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 240 0 0; 3 240 240 0; 4 480 240 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 TABLE ST PIPE OD 12 ID 10.255
PRINT MEMBER PROPERTIES
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.64e+07
POISSON 0.3
ALPHA 7.26744e-06
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
* DONOT PROVIDE POISSON'S RATIO, NO SHEAR DEFORMATION CONSIDERED
SUPPORTS
1 4 FIXED
LOAD 1
TEMPERATURE LOAD
1 TO 3 TEMP 430
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES LIST 1
FINISH

---

**STAAD Output**

<table>
<thead>
<tr>
<th>SUPPORT REACTIONS -UNIT POUN INCH</th>
<th>STRUCTURE TYPE = PLANE</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM Z</td>
<td></td>
</tr>
<tr>
<td>1  1  8949.43  7729.42  0.00  0.00  0.00  781127.94</td>
<td></td>
</tr>
<tr>
<td>4  1 -8949.43  -7729.42  0.00  0.00  0.00  781127.94</td>
<td></td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

34. PRINT MEMBER FORCES LIST 1
MEMBER FORCES LIST 1
 :A FIXED-FIXED ZEE SHAPED PLANE BENT BEAM  -- PAGE NO. 5

MEMBER END FORCES  STRUCTURE TYPE = PLANE

----------------------
ALL UNITS ARE -- POUN INCH   (LOCAL)
MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
1  1  1  8949.43  7729.42  0.00  0.00  0.00  781127.94
2 -8949.43 -7729.42  0.00  0.00  0.00 1073932.12
```

---

**STAAD.Pro** 3243  **User Manual**
V.02 Trusses

V. Axial Stress on a Truss Model

To find member stress due to a joint load in a space truss using static analysis.

Reference


Problem

A 50 lb load is supported by three bars which are pinned to a ceiling as shown. Determine the stress, $\sigma$, in each bar.

Area of each bar = 1 in$^2$, $E = 30 \times 10^6$ psi

![Figure 364: Space truss](image)

$P = 50$ lbs

Comparison

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{AD}$</td>
<td>31.2</td>
<td>31.2</td>
<td>none</td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.02 Trusses

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\sigma_{BD})</td>
<td>10.4</td>
<td>10.4</td>
<td>none</td>
</tr>
<tr>
<td>(\sigma_{CD})</td>
<td>22.9</td>
<td>22.9</td>
<td>none</td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Stress on a Truss Model.STD is typically installed with the program.

**STAAD TRUSS :A SPACE TRUSS**

**START JOB INFORMATION**

**ENGINEER DATE 18-Sep-18**

**END JOB INFORMATION**

* REF: "VECTOR MECHANICS FOR ENGINEERS, STATICS & DYNAMICS",
  * MCGRAW HILL BOOK CO., INC., NEW YORK, 1962
  * PROBLEM 2.70, PAGE 47.

UNIT FEET POUND

JOINT COORDINATES

1 0 0 0; 2 8 0 0; 3 0 6 0; 4 4 2 6;

MEMBER INCIDENCES

1 1 4; 2 2 4; 3 3 4;

UNIT INCHES POUND

MEMBER PROPERTY AMERICAN

1 TO 3 PRIS AX 1

**DEFINE MATERIAL START**

**ISOTROPIC MATERIAL1**

E 3e+07

POISSON 0.290909

**END DEFINE MATERIAL**

**CONSTANTS**

**MATERIAL MATERIAL1 ALL**

**SUPPORTS**

1 TO 3 PINNED

**LOAD 1 WEIGHT**

**JOINT LOAD**

4 FZ 50

**PERFORM ANALYSIS**

**PRINT MEMBER STRESSES**

**FINISH**

### STAAD Output

**ALL UNITS ARE POUN/SQ INCH**

<table>
<thead>
<tr>
<th>MEMB</th>
<th>LD</th>
<th>SECT</th>
<th>AXIAL</th>
<th>BEND-Y</th>
<th>BEND-Z</th>
<th>COMBINED</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1.0</td>
<td>10.4 T</td>
<td>0.0</td>
<td>0.0</td>
<td>10.4</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>1.0</td>
<td>10.4 T</td>
<td>0.0</td>
<td>0.0</td>
<td>10.4</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.0</td>
<td>31.2 T</td>
<td>0.0</td>
<td>0.0</td>
<td>31.2</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>1.0</td>
<td>31.2 T</td>
<td>0.0</td>
<td>0.0</td>
<td>31.2</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>
V. Axial Force on a Cable

To find member force due to a member load in a plane articulate structure.

Reference
(Original data is in US Customary Units)

Problem
Find the tensile stress in the cable. The cross-sectional area of the cable is 967.74 mm² with an E of 137.895 GPa. The timber beam 1-3 is 304.8 mm x 304.8 mm in section, with E = 11.03161 GPa. Each member of the steel cantilever truss has a cross-sectional area of 2,580.64 mm², and E of 206.8427 GPa.

![Figure 365: Plane articulate truss](image)

Comparison

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cable, 3-4</td>
<td>22.7</td>
<td>22.58</td>
<td>0.6% (negligible)</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Force on a Cable.STD is typically installed with the program.

STAAD PLANE :A PLANE ARTICULATE STRUCTURE
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE: INDETERMINATE STRURAL ANALYSIS, KINNEY, 1957,
* ADISON-WESLEY PUBLISHING COMPANY, PAGE 275, PROBLEM 6-19.
*

UNIT METER NEWTON
JOIN COORDINATES
1 0 0 0; 2 3.048 0 0; 3 4.572 0 0; 4 4.572 2.4384 0; 5 6.4008 4.8768 0;
6 6.4008 2.4384 0; 7 8.2296 4.8768 0; 8 8.2296 2.4384 0;
9 10.0586 4.8768 0; 10 10.0586 2.4384 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 6; 5 6 8; 6 8 10; 7 4 5; 8 6 5; 9 6 7; 10 8 7;
11 8 9; 12 5 7; 13 7 9;
MEMBER PROPERTY AMERICAN
1 2 PRIS YD 0.3048 ZD 0.3048
3 PRIS AX 0.00096774 IZ 4.16231e-07
4 TO 13 PRIS AX 0.00258064 IZ 4.16231e-07
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1.10316e+07
POISSON 0.22857
ISOTROPIC MATERIAL2
E 1.37895e+08
POISSON 0.261539
ISOTROPIC MATERIAL3
E 2.06843e+08
POISSON 0.290909
END DEFINE MATERIAL
UNIT METER NEWTON
CONSTANTS
MATERIAL MATERIAL1 MEMB 1 2
MATERIAL MATERIAL2 MEMB 3
MATERIAL MATERIAL3 MEMB 4 TO 13
MEMBER RELEASE
3 TO 13 START MZ
SUPPORTS
1 9 10 FIXED
LOAD 1
JOINT LOAD
2 FY -44482.2
PERFORM ANALYSIS
UNIT METER KN
PRINT MEMBER FORCES LIST 3
FINISH
V. Axial Force in a 2D Plane Frame 1

To find member forces due to joint loads in a plane truss.

Reference


Problem

Compute the bar forces in the bars a, b, c, d, e of the truss due to the loads shown.

![Figure 366: Plane truss](image)

Comparison

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>-202.13</td>
<td>-202.13</td>
<td>none</td>
</tr>
<tr>
<td>b</td>
<td>-185.76</td>
<td>-190.28</td>
<td>2.4% (negligible)</td>
</tr>
</tbody>
</table>
### Result Type

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>-22.42</td>
<td>-23.42</td>
<td>4.5%</td>
</tr>
<tr>
<td>d</td>
<td>266.23</td>
<td>269.52</td>
<td>1.2% (negligible)</td>
</tr>
<tr>
<td>e</td>
<td>119.61</td>
<td>117.08</td>
<td>2.1% (negligible)</td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Force in a 2D Plane Frame 1.STD is typically installed with the program.

```
STAAD TRUSS :A PLANE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE: ELEMENTARY STRUCTURAL ANALYSIS, NORIS AND WILBUR,
  2nd EDITION, McGRAW-HILL BOOK COMPANY, PAGE 159,
  PROBLEM 4.3 (ORIGINAL DATA IN US CUSTOMARY UNITS)

UNIT METER NEWTON

JOIN COORDINATES
1 0 0 0; 2 7.62 0 0; 3 7.62 9.144 0; 4 15.24 0 0; 5 15.24 4.572 0;
6 15.24 9.144 0; 7 22.86 0 0; 8 22.86 12.192 0; 9 30.48 0 0;
10 30.48 4.572 0; 11 30.48 9.144 0; 12 38.1 0 0; 13 38.1 9.144 0;
14 45.72 0 0;

MEMBER INCIDENCES
1 2 1 5; 3 1 3; 4 2 3; 5 2 4; 6 5 3; 7 3 6; 8 4 7; 9 7 5; 10 6 8;
11 7 9; 12 7 10; 13 11 8; 14 9 12; 15 10 13; 16 11 13; 17 12 14;
18 14 10; 19 12 13; 20 14 13; 21 4 5; 22 5 6; 23 7 8; 24 9 10; 25 10 11;

MEMBER PROPERTY AMERICAN
1 TO 25 PRIS AX 0.00129032

UNIT METER KN

DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.06843e+08
POISSON 0.290909

END DEFINE MATERIAL

UNIT METER NEWTON

CONSTANTS
MATERIAL MATERIAL1 ALL

MEMBER TRUSS
1 TO 25

SUPPORTS
1 PINNED
14 FIXED BUT FX

LOAD 1 VERTICAL JOINT LOADS

PERFORM ANALYSIS
```
V. Axial Forces on a 3D Space Model

To find support reactions and member forces due to a joint load in a space truss.

Reference

Problem
The space truss is supported by the six reactions shown. If a horizontal 2,700 N load is applied at A, determine the reactions and the force in each member.
Comparison

Table 384: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>B_v</td>
<td>0</td>
<td>0</td>
<td>none</td>
</tr>
<tr>
<td>B_z</td>
<td>2,700</td>
<td>2,700</td>
<td>none</td>
</tr>
<tr>
<td>C_x</td>
<td>1,800</td>
<td>1,800</td>
<td>none</td>
</tr>
<tr>
<td>C_v</td>
<td>3,375</td>
<td>3,375</td>
<td>none</td>
</tr>
<tr>
<td>D_x</td>
<td>1,800</td>
<td>1,800</td>
<td>none</td>
</tr>
<tr>
<td>D_v</td>
<td>3,375</td>
<td>3,375</td>
<td>none</td>
</tr>
</tbody>
</table>

Table 385: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>AB</td>
<td>0</td>
<td>0</td>
<td>none</td>
</tr>
<tr>
<td>AC</td>
<td>4,275</td>
<td>4,275</td>
<td>none</td>
</tr>
<tr>
<td>AD</td>
<td>4,275</td>
<td>4,275</td>
<td>none</td>
</tr>
<tr>
<td>BC</td>
<td>4,270</td>
<td>4,269</td>
<td>negligible</td>
</tr>
<tr>
<td>Result Type</td>
<td>Theory</td>
<td>STAAD.Pro</td>
<td>Difference</td>
</tr>
<tr>
<td>-------------</td>
<td>--------</td>
<td>-----------</td>
<td>------------</td>
</tr>
<tr>
<td>CD</td>
<td>0</td>
<td>0</td>
<td>none</td>
</tr>
<tr>
<td>BD</td>
<td>4,270</td>
<td>4,269</td>
<td>negligible</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Forces on a 3D Space Model.STD is typically installed with the program.

STAAD TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE: VECTOR MECHANICS FOR ENGINEERS - STATICS, BEER AND JOHNSTON, 4th EDITION, McGRAW-HILL BOOK CO., PAGE 216, PROBLEM 6.20.

UNIT METER NEWTON
JOINT COORDINATES
1 1 0 0.6; 2 1 0 -0.6; 3 -0.8 0 0; 4 0 1.5 0;
MEMBER INCIDENCES
1 1 4; 2 2 4; 3 3 4; 4 1 2; 5 2 3; 6 3 1;
MEMBER PROPERTY AMERICAN
1 TO 6 PRIS AX 0.001
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+11
POISSON 0.29542
END DEFINE MATERIAL CONSTANTS
MATERIAL MATERIAL1 ALL
MEMBER TRUSS
1 TO 6
SUPPORTS
1 FIXED BUT FZ
2 FIXED BUT FZ
3 FIXED BUT FX
LOAD 1 HORIZONTAL LOAD
JOINT LOAD
4 FZ 2700
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
FINISH

STAAD Output

SUPPORT REACTION
STAAD TRUSS
V. Reactions in a 2D Truss Model 1

To find support reactions due to joint loads in a plane truss.

Reference


Problem

Find the vertical support reactions of the truss.

\[ E = 30,000.0 \text{ ksi} \]

\[ A = 100 \text{ in}^2 \]

Loads as shown.
Comparison

Table 386: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_1$ (kips)</td>
<td>-76.7</td>
<td>-76.67</td>
<td>none</td>
</tr>
<tr>
<td>$R_4$ (kips)</td>
<td>346.7</td>
<td>346.67</td>
<td>none</td>
</tr>
<tr>
<td>$R_9$ (kips)</td>
<td>30</td>
<td>30</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Reactions in a 2D Truss Model 1.STD is typically installed with the program.

STAAD TRUSS : A PLANE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MCCORMAC, J.C., "STRUCTURAL ANALYSIS", INTEXT
* EDUCATIONAL PUBLISHERS, NEW YORK, 3RD EDITION, 1975.
*
INPUT WIDTH 72
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0; 3 60 0 0; 4 90 0 0; 5 120 0 0; 6 150 0 0; 7 180 0 0;
8 210 0 0; 9 240 0 0; 10 0 20 0; 11 30 20 0; 12 60 20 0; 13 90 20 0;
14 120 20 0; 15 150 20 0; 16 180 20 0; 17 210 20 0; 18 240 20 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 10 11;
V. Reactions in a 2D Truss Model 2

To find support reactions due to joint loads in a plane truss.

Reference


Problem

Find the vertical and horizontal reactions at the supports of the truss.

\[ E = 30,000.0 \text{ ksi} \]

\[ A = 100 \text{ in}^2 \]

Loads as shown.
Comparison

Table 387: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal, R₁ (kips)</td>
<td>11.5</td>
<td>11.49</td>
<td>none</td>
</tr>
<tr>
<td>Vertical, R₁ (kips)</td>
<td>34.3</td>
<td>34.29</td>
<td>none</td>
</tr>
<tr>
<td>Horizontal, R₂ (kips)</td>
<td>-31.7</td>
<td>-31.67</td>
<td>none</td>
</tr>
<tr>
<td>Vertical, R₂ (kips)</td>
<td>36.0</td>
<td>36.0</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Reactions in a 2D Truss Model 2.STD is typically installed with the program.

STAAD TRUSS :A PLANE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* REFERENCE: MCCORMAC, J.C.,"STRUCTURAL ANALYSIS", INTEXT
* EDUCATIONAL PUBLISHERS, NEW YORK, 3RD EDITION, 1975.
INPUT WIDTH 72
UNIT FEET KIP
JOIN COORDINATES
1 0 0 0; 2 0 60 0; 3 12.5 60 0; 4 0 15 0; 5 0 30 0; 6 0 45 0;
7 9.375 45 0; 8 6.25 30 0; 9 3.125 15 0; 10 50 90 0; 11 100 90 0;
12 50 75 0; 13 25 75 0; 14 66.6667 90 0; 15 83.3333 90 0; 16 83.3333 85 0;
17 66.6667 80 0; 18 31.25 67.5 0; 19 200 0 0; 20 200 60 0; 21 187.5 60 0;
22 200 15 0; 23 200 30 0; 24 200 45 0; 25 190.625 45 0; 26 193.75 30 0;
27 196.875 15 0; 28 150 90 0; 29 150 75 0; 30 175 75 0; 31 133.3333 90 0;
32 116.6667 90 0; 33 116.6667 85 0; 34 133.3333 80 0; 35 168.75 67.5 0;
UNIT INCHES KIP
MEMBER INCIDENCES
1 1 4; 2 2 3; 3 3 7; 4 4 5; 5 5 6; 6 6 2; 7 7 8; 8 8 9; 9 9 1; 10 4 9;
11 9 5; 12 5 8; 13 8 6; 14 6 7; 15 7 2; 16 2 13; 17 10 14; 18 11 16;
19 12 18; 20 13 10; 21 14 15; 22 15 11; 23 16 17; 24 17 12; 25 18 3;
26 3 13; 27 13 18; 28 13 12; 29 12 10; 30 10 17; 31 17 14; 32 14 16;
33 16 15; 34 19 22; 35 20 21; 36 21 25; 37 22 23; 38 23 24; 39 24 20;
40 25 26; 41 26 27; 42 27 19; 43 22 23; 44 27 23; 45 23 26; 46 26 24;
47 24 25; 48 25 20; 49 20 30; 50 28 31; 51 11 33; 52 29 35; 53 30 28;
54 31 32; 55 32 11; 56 33 34; 57 34 29; 58 35 21; 59 21 30; 60 30 35;
61 30 29; 62 29 28; 63 28 34; 64 34 31; 65 31 33; 66 33 32;
UNIT FEET KIP
MEMBER PROPERTY AMERICAN
1 TO 66 PRIS AX 100
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.32e+06
POISSON 0.290909
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 19 PINNED
UNIT FEET KIP
LOAD 1 JOINT LOADS AT SPECIFIC NODES
JOIN LOAD
11 14 15 31 32 FY -10
10 28 FY -5
4 TO 6 FX 4
2 FX 2
2 10 FX 1.5435
13 FX 3.087
2 10 FY -2.5725
13 FY -5.145
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH

STAAD Output
SUPPORT REACTIONS -UNIT KIP FEET
-----------------------------
JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM-Z
V. Reactions in a 2D Truss Model 3

Find the support reactions due to a joint load in a plane truss.

Reference

Problem
Determine the horizontal reaction at support 4 of the system.

L = 50 in.
P = 10 kips

\[ \begin{array}{cccccccc}
1 & 1 & 11.49 & 34.29 & 0.00 & 0.00 & 0.00 & 0.00 \\
19 & 1 & -31.67 & 36.00 & 0.00 & 0.00 & 0.00 & 0.00 \\
\end{array} \]

Comparison

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>R_4 (kips)</td>
<td>8.77</td>
<td>8.77</td>
<td>none</td>
</tr>
</tbody>
</table>

Figure 370: Plane truss
**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Reactions in a 2D Truss Model 3.STD` is typically installed with the program.

```
STAAD TRUSS : REACTION IN TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
*  REFERENCE `STRENGTH OF MATERIALS' PART-1 BY S. TIMOSHENKO
*  PAGE 346 PROBLEM NO. 3. THE ANSWER IS REACTION = 0.877P.
*  THEREFORE IF P=10, REACTION = 8.77
*
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 150 100 0; 3 150 50 0; 4 300 0 0;
MEMBER INCIDENCES
1 1 2; 2 1 3; 3 2 3; 4 2 4; 5 3 4;
MEMBER PROPERTY AMERICAN
1 4 PRIS AX 5
2 5 PRIS AX 3
3 PRIS AX 2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 PINNED
LOAD 1
JOINT LOAD
2 FY -10
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

**STAAD Output**

```
SUPPORT REACTIONS -UNIT KIP INCH STRUCTURE TYPE = TRUSS
-----------------  
JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM Z
1 1 8.77 5.00 0.00 0.00 0.00
4 1 -8.77 5.00 0.00 0.00 0.00
```

---

**V. Deflections in a 2D Truss Model**

Find the joint deflection due to joint loads in a plane truss.
Reference


Problem

Determine the vertical deflection at point 5 of plane truss structure shown in the figure.

![Truss Model](image)

*Figure 371: Plane truss model*

\[ P = 20 \text{ kip} \]
\[ L = 15 \text{ ft} \]

Truss width = 4 spaces at 15 ft = 60 ft

Truss height = 15 ft

\[ A_{X1-4} = 1 \text{ in}^2, \ A_{X5-6} = 2 \text{ in}^2, \ A_{X7-8} = 1.5 \text{ in}^2, \]
\[ A_{X9-11} = 3 \text{ in}^2, \ A_{X12-13} = 4 \text{ in}^2 \]

\[ E = 30E3 \text{ ksi} \]

Comparison

<table>
<thead>
<tr>
<th>Table 389: Comparison of results</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Result Type</strong></td>
</tr>
<tr>
<td>( \delta_5 ) (in.)</td>
</tr>
</tbody>
</table>
**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Deflections in a 2D Truss Model.STD is typically installed with the program.

**STAAD TRUSS : DEFLECTION IN TRUSS**

**START JOB INFORMATION**

**ENGINEER DATE** 18-Sep-18

**END JOB INFORMATION**

* 
*  REFERENCE 'STRUCTURAL ANALYSIS' BY JACK McCORMACK, PAGE
*  271 EXAMPLE 18-2. ANSWER - Y-DISP AT JOINT 5 = 2.63 INCH
* 

**UNIT FEET KIP**

**JOINT COORDINATES**

1 0 0 0; 2 15 0 0; 3 30 0 0; 4 45 0 0; 5 60 0 0; 6 15 7.5 0; 7 30 15 0; 8 45 7.5 0;

**MEMBER INCIDENCES**

1 2 6; 2 3 4; 3 4 8; 4 4 5; 5 1 2; 6 2 3; 7 3 6; 8 3 8; 9 3 7; 10 1 6; 11 5 8; 12 6 7; 13 7 8;

**UNIT INCHES KIP**

**MEMBER PROPERTY AMERICAN**

1 TO 4 PRIS AX 1
5 6 PRIS AX 2
7 8 PRIS AX 1.5
9 TO 11 PRIS AX 3
12 13 PRIS AX 4

**DEFINE MATERIAL START**

**ISOTROPIC MATERIAL1**

E 30000
POISSON 0.3

**END DEFINE MATERIAL**

**CONSTANTS**

**MATERIAL MATERIAL1 ALL**

**SUPPORTS**

1 PINNED
3 FIXED BUT FX MZ

**LOAD 1 VERTICAL LOAD**

**JOINT LOAD**

2 4 5 FY -20

**PERFORM ANALYSIS**

**PRINT JOINT DISPLACEMENTS**

**FINISH**

---

**STAAD Output**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-0.12000</td>
<td>0.18000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-0.24000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-0.48000</td>
<td>-0.89516</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>
V. Stress in a 2D Truss Model

Find the joint deflection and member stress due to a joint load in a plane truss.

Reference

Problem
Determine the vertical deflection at point A and the member stresses

\[ A_X = 0.5 \text{ in}^2 \]
\[ E = 30\text{E6 psi} \]
\[ P = 5000 \text{ lbf} \]
\[ L = 180 \text{ in.} \]
\[ \text{angle} = 30^\circ \]

![Figure 372: Model of two member truss](image)

Comparison

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_A ) (psi)</td>
<td>10,000.</td>
<td>10,000.</td>
<td>none</td>
</tr>
<tr>
<td>( \delta_A ) (in)</td>
<td>0.12</td>
<td>0.12</td>
<td>none</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Stress in a 2D Truss Model.STD is typically installed with the program.

STAAD TRUSS : STRESS IN TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* THIS EXAMPLE IS TAKEN FROM 'STRENGTH OF MATERIALS'
* (PART 1) BY TIMOSHENKO, PAGE 10 PROB 2.
* THE ANSWER IN THE BOOK, DEFLECTION = 0.12 INCH
* AND STRESS =10000 PSI
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 155.885 -90 0; 3 311.769 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 0.5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.15
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 3 PINNED
LOAD 1 VERT LOAD
JOINT LOAD
2 FY -5000
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
PRINT MEMBER STRESSES
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>JOINT DISPLACEMENT (INCH RADIANS)</th>
<th>STRUCTURE TYPE = TRUSS</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD</td>
<td>X-TRANS</td>
</tr>
<tr>
<td>1 1 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000</td>
<td></td>
</tr>
<tr>
<td>2 1 0.00000 -0.12000 0.00000 0.00000 0.00000 0.00000</td>
<td></td>
</tr>
<tr>
<td>3 1 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000</td>
<td></td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

32. PRINT MEMBER STRESSES
MEMBER STRESSES
: STRESS IN TRUSS
--- PAGE NO.

4
MEMBER STRESSES
V. Axial Forces in a Plane Frame 2

To find member forces due to a thermal load in a plane truss.

Reference


Problem

A symmetric, three-bar truss ABCD undergoes a temperature increase of 20°C in the two outer bars and 70°C in the middle bar. Calculate the forces $F_1$ and $F_2$ in the bars.

$$E = 200 \text{ GPa}$$

$$\alpha = 14 \times 10^{-6} / ^\circ\text{C}$$

$$A = 900 \text{ mm}^2$$

![Figure 373: Plane truss subject to differential thermal loading](image)

Comparison

**Table 391: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_1$</td>
<td>22,100</td>
<td>22,143</td>
<td>0.2% (negligible)</td>
</tr>
</tbody>
</table>
### STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Forces in a Plane Frame 2.STD` is typically installed with the program.

```
STAAD TRUSS : A PLANE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION

* * REFERENCES: MECHANICS OF MATERIALS, GERE AND TIMOSHENKO, 2ND EDITION
* * PWS ENGINEERING, PAGE 121, PROBLEM 2.6-23

UNIT METER NEWTON
JOINT COORDINATES
1 -2.12132 2.12132 0; 2 0 2.12132 0; 3 2.12132 2.12132 0; 4 0 0 0;
MEMBER INCIDENCES
1 1 4; 2 2 4; 3 3 4;
UNIT MMS NEWTON
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 900
UNIT METER NEWTON
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+11
POISSON 0.29542
ALPHA 1.4e-05
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 3 PINNED
LOAD 1 TEMPERATURE LOAD
TEMPERATURE LOAD
1 3 TEMP 20 800
2 TEMP 70
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH
```
V. Roof Truss Axial Forces

To find the forces in the members of a truss structure due to fabrication defect in the length of one bar which is 0.75 cm too short.

Reference

Problem
Structure to be solved as a truss. To achieve this in the STAAD model, instead of declaring them truss members, define them as frame members, and release MZ at both ends of all members. In the above figure, the area of cross section of the members is as follows.

1 TO 4 = 20 cm²
5, 6 = 24 cm²
7, 8 = 30 cm²
9, 10 = 15 cm²
Comparison

Table 392: Comparison of results

<table>
<thead>
<tr>
<th>Member Number</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>280.56 (C)</td>
<td>-280.57</td>
<td>none</td>
</tr>
<tr>
<td>2</td>
<td>50.86 (T)</td>
<td>50.86</td>
<td>none</td>
</tr>
<tr>
<td>3</td>
<td>127.99 (C)</td>
<td>-127.99</td>
<td>none</td>
</tr>
<tr>
<td>4</td>
<td>101.72 (T)</td>
<td>101.72</td>
<td>none</td>
</tr>
<tr>
<td>5</td>
<td>134.14 (C)</td>
<td>-134.14</td>
<td>none</td>
</tr>
<tr>
<td>6</td>
<td>57.85 (C)</td>
<td>-57.85</td>
<td>none</td>
</tr>
<tr>
<td>7</td>
<td>63.57 (C)</td>
<td>-63.57</td>
<td>none</td>
</tr>
<tr>
<td>8</td>
<td>127.14 (C)</td>
<td>-127.15</td>
<td>none</td>
</tr>
<tr>
<td>9</td>
<td>287.13 (T)</td>
<td>287.13</td>
<td>none</td>
</tr>
<tr>
<td>10</td>
<td>223.56 (T)</td>
<td>223.56</td>
<td>none</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Roof Truss Axial Forces.STD is typically installed with the program.

STAAD SPACE EFFECT OF FABRICATION DEFECT - PRESTRAIN

START JOB INFORMATION
ENGINEER DATE 10-Oct-17
END JOB INFORMATION

* REFERENCE: INTERMEDIATE STRUCTURAL ANALYSIS, C.K.WANG
* INTERNATIONAL STUDENT EDITION, 1983, MCGRAW HILL
* SECTION 10.13, PAGE 387

UNIT METER KN

JOINT COORDINATES
1 2.4 1.8 0; 2 4.8 1.8 0; 3 0 0 0; 4 2.4 0 0; 5 4.8 0 0; 6 7.2 0 0;

MEMBER INCIDENCES
1 1 2; 2 3 4; 3 4 5; 4 5 6; 5 4 1; 6 5 2; 7 3 1; 8 2 6; 9 1 5; 10 4 2;

UNIT CM KN

MEMBER PROPERTY AMERICAN
1 TO 4 PRIS AX 20 IX 0.01 IY 0.01 IZ 0.01
5 6 PRIS AX 24 IX 0.01 IY 0.01 IZ 0.01
7 8 PRIS AX 30 IX 0.01 IY 0.01 IZ 0.01
9 10 PRIS AX 15 IX 0.01 IY 0.01 IZ 0.01

UNIT CM NEWTON

DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+07
POISSON 0.3
END DEFINE MATERIAL

UNIT CM KN

CONSTANTS

MATERIAL MATERIAL1 ALL
* ALL MOMENTS RELEASED TO SIMULATE TRUSS ACTION

MEMBER RELEASE
5 6 9 10 START MZ
5 6 9 10 END MZ
*1 TO 10 START MZ
*1 TO 10 END MZ

SUPPORTS
3 PINNED
5 6 FIXED BUT FX MZ

LOAD 1
TEMPERATURE LOAD
9 STRAIN -0.75
PERFORM ANALYSIS

UNIT CM NEWTON
PRINT MEMBER FORCES
FINISH
**STAAD Output**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>280565.47</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-4.05</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>3</td>
<td>-50858.06</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.38</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>1</td>
<td>127991.16</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-4.02</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>5</td>
<td>-101716.24</td>
<td>0.01</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2.67</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>1</td>
<td>134136.97</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>5</td>
<td>57849.78</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>3</td>
<td>63572.59</td>
<td>0.01</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>4.05</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>2</td>
<td>127145.30</td>
<td>0.01</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2.71</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>1</td>
<td>287134.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>4</td>
<td>-223561.55</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**V.03 Frames**

**V. 2D Portal Reactions 1**

To find section properties, member forces and support reactions for a 1x1 bay plane frame with members of rectangular section.

**Reference**


**Problem**

The frame supports a concentrated load at middle of the horizontal member. Verify the internally calculated section properties, support reactions and bending moments at the ends of the horizontal member. Columns are square 2" x 2"; beams are rectangular with b = 2" and h = 4".

\[
\begin{align*}
P &= 1,000 \text{ lb} \\
h &= 100 \text{ in.} \\
l &= 120 \text{ in.}
\end{align*}
\]
E = 30\times(10)^6 \text{ psi}

![Symmetric portal frame](image)

**Figure 375: Symmetric portal frame**

**Calculations**

From the reference:

\[
M = \frac{Pl}{8} \left(1 + \frac{1}{\frac{2}{3} \frac{h}{I_b}} \right) = \frac{1,000(120)}{8} \left(1 + \frac{1}{\frac{2}{3} \frac{100}{1.333}} \right) = 2,754\text{in-lb}
\]

**Comparison**

**Table 393: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Column Cross Section</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(A_x) (in.(^2))</td>
<td>4.0</td>
<td>4.0</td>
<td>none</td>
</tr>
<tr>
<td>(I_x) (in.(^4))</td>
<td>2.25</td>
<td>2.25</td>
<td>none</td>
</tr>
<tr>
<td>(I_y) (in.(^4))</td>
<td>1.333</td>
<td>1.33</td>
<td>none</td>
</tr>
<tr>
<td>(I_z) (in.(^4))</td>
<td>1.333</td>
<td>1.33</td>
<td>none</td>
</tr>
<tr>
<td><strong>Beam Cross Section</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(A_x) (in.(^2))</td>
<td>8.0</td>
<td>8.0</td>
<td>none</td>
</tr>
<tr>
<td>(I_x) (in.(^4))</td>
<td>7.324</td>
<td>7.32</td>
<td>none</td>
</tr>
<tr>
<td>(I_y) (in.(^4))</td>
<td>2.667</td>
<td>2.67</td>
<td>none</td>
</tr>
<tr>
<td>(I_z) (in.(^4))</td>
<td>10.667</td>
<td>10.67</td>
<td>none</td>
</tr>
<tr>
<td>(R_y) (lb)</td>
<td>500</td>
<td>500</td>
<td>none</td>
</tr>
<tr>
<td>Result Type</td>
<td>Theory</td>
<td>STAAD.Pro</td>
<td>Difference</td>
</tr>
<tr>
<td>-------------</td>
<td>---------</td>
<td>-----------</td>
<td>------------</td>
</tr>
<tr>
<td>$R_x$ (lb)</td>
<td>27.55</td>
<td>27.54</td>
<td>none</td>
</tr>
<tr>
<td>$M$(in·lb)</td>
<td>2,754.97</td>
<td>2,754.35</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2D Portal Reactions 1.STD is typically installed with the program.

STAAD PLANE : A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*  
* REFERENCE: TIMOSHENKO, S., STRENGTH OF MATERIALS, PART 1,  
* 2ND EDITION, D. VAN NOSTRAND CO., 1940,  
* PAGES 188 THRU 191.  
*  
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 0 100 0; 3 60 100 0; 4 120 100 0; 5 120 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5;
MEMBER PROPERTY AMERICAN
1 4 PRIS YD 2 ZD 2 2 3 PRIS YD 4 ZD 2
PRINT MEMBER PROPERTIES
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL CONSTANTS
MATERIAL MATERIAL1 ALL SUPPORTS
1 5 PINNED
LOAD 1 MID SPAN
JOINT LOAD
3 FY 1000
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES LIST 2 3
FINISH

**STAAD Output**

*  ****************************************************  
*  ****************************************************  
*  Verification Examples  
*  V.03 Frames  
*  STAAD.Pro  
*  3271  
*  User Manual  
*  ******************
1. STAAD PLANE: A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION

INPUT FILE: 2D Portal Reactions 1.STD

2. START JOB INFORMATION

3. ENGINEER DATE 17-SEP-18

4. END JOB INFORMATION

5. *

6. * REFERENCE: TIMOSHENKO, S., STRENGTH OF MATERIALS, PART 1,
   2ND EDITION, D. VAN NOSTRAND CO., 1940,
   PAGES 188 THRU 191.

7. *

8. *

9. *

10. UNIT INCHES POUND

11. JOINT COORDINATES

12. 1 0 0 0; 2 0 100 0; 3 60 100 0; 4 120 100 0; 5 120 0 0

13. MEMBER INCIDENCES

14. 1 1 2; 2 2 3; 3 3 4; 4 4 5

15. MEMBER PROPERTY AMERICAN

16. 1 4 PRIS YD 2 ZD 2

17. 2 3 PRIS YD 4 ZD 2

18. PRINT MEMBER PROPERTIES

MEMBER PROPERTIES

1. STAAD PLANE: A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION

MEMBER PROPERTIES. UNIT - INCH

-----------------
MEMB PROFILE AX/ IZ/ IY/ IX/ AY AZ SZ SY

1 PRISMATIC 4.00 1.33 1.33 2.25 3.40 3.40 1.33 1.33

2 PRISMATIC 8.00 10.67 2.67 7.32 6.80 6.80 5.33 2.67

3 PRISMATIC 8.00 10.67 2.67 7.32 6.80 6.80 5.33 2.67

4 PRISMATIC 4.00 1.33 1.33 2.25 3.40 3.40 1.33 1.33

************ END OF DATA FROM INTERNAL STORAGE ************

19. DEFINE MATERIAL START

20. ISOTROPIC MATERIAL1

21. E 3E+07

22. POISSON 0.290909

23. END DEFINE MATERIAL

24. CONSTANTS

25. MATERIAL MATERIAL1 ALL

26. SUPPORTS

27. 1 5 PINNED

28. LOAD 1 MID SPAN

29. JOINT LOAD

30. 3 FY 1000

31. PERFORM ANALYSIS

** PROBLEM STATISTICS **
V. 3x2 Plane Frame Moments

To find the bending moments due to lateral joint loads in a 3x2 bay plane frame.
Reference

Problem
Determine the bending moments in the members of frame.

\[
E = 30,000 \text{ ksi} \\
I_{AE} = I_{EI} = 240 \text{ in}^4 \\
I_{BF} = I_{FJ} = 480 \text{ in}^4 \\
I_{CG} = I_{GK} = 600 \text{ in}^4 \\
I_{DH} = I_{HL} = 360 \text{ in}^4 \\
I_{EF} = I_{IJ} = 600 \text{ in}^4 \\
I_{FG} = I_{JK} = 1,200 \text{ in}^4 \\
I_{GH} = I_{KL} = 1,800 \text{ in}^4
\]

*Figure 376: 3x2 bay plane frame*
## Comparison

### Table 394: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending in member (ft-kips)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>AE</td>
<td>29.6</td>
<td>29.80</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>EA</td>
<td>25.1</td>
<td>25.21</td>
<td>none</td>
</tr>
<tr>
<td>EI</td>
<td>6.5</td>
<td>6.44</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>IE</td>
<td>10.6</td>
<td>10.48</td>
<td>1.1%</td>
</tr>
<tr>
<td>BF</td>
<td>60.9</td>
<td>61.03</td>
<td>none</td>
</tr>
<tr>
<td>FB</td>
<td>53.7</td>
<td>53.69</td>
<td>none</td>
</tr>
<tr>
<td>FJ</td>
<td>18.3</td>
<td>18.50</td>
<td>1.1%</td>
</tr>
<tr>
<td>JF</td>
<td>24.8</td>
<td>24.84</td>
<td>none</td>
</tr>
<tr>
<td>CG</td>
<td>76.8</td>
<td>76.70</td>
<td>none</td>
</tr>
<tr>
<td>GC</td>
<td>68.3</td>
<td>68.24</td>
<td>none</td>
</tr>
<tr>
<td>GK</td>
<td>25.2</td>
<td>25.25</td>
<td>none</td>
</tr>
<tr>
<td>KG</td>
<td>32.4</td>
<td>32.52</td>
<td>none</td>
</tr>
<tr>
<td>DH</td>
<td>45.6</td>
<td>45.45</td>
<td>none</td>
</tr>
<tr>
<td>HD</td>
<td>40.1</td>
<td>39.87</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>HL</td>
<td>13.5</td>
<td>13.58</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>LH</td>
<td>18.5</td>
<td>18.39</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>EF</td>
<td>31.6</td>
<td>-31.65</td>
<td>none</td>
</tr>
<tr>
<td>FE</td>
<td>29.5</td>
<td>-29.35</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>IJ</td>
<td>10.6</td>
<td>-10.48</td>
<td>1.1%</td>
</tr>
<tr>
<td>JI</td>
<td>10.0</td>
<td>-9.79</td>
<td>2.1%</td>
</tr>
<tr>
<td>FG</td>
<td>42.5</td>
<td>-42.84</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>GF</td>
<td>41.3</td>
<td>-41.71</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>JK</td>
<td>14.8</td>
<td>-15.05</td>
<td>1.7%</td>
</tr>
<tr>
<td>Result Type</td>
<td>Theory</td>
<td>STAAD.Pro</td>
<td>Difference</td>
</tr>
<tr>
<td>-------------</td>
<td>--------</td>
<td>-----------</td>
<td>------------</td>
</tr>
<tr>
<td>KJ</td>
<td>14.4</td>
<td>-14.71</td>
<td>2.2%</td>
</tr>
<tr>
<td>GH</td>
<td>52.2</td>
<td>-51.78</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>HG</td>
<td>53.6</td>
<td>-53.45</td>
<td>none</td>
</tr>
<tr>
<td>KL</td>
<td>18.0</td>
<td>-17.81</td>
<td>1.1%</td>
</tr>
<tr>
<td>LK</td>
<td>18.5</td>
<td>-18.39</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\3x2 Plane Frame Moments.STD` is typically installed with the program.

STAAD PLANE :A 3X2 BAY PLANE FRAME
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION

* REFERENCE: ELEMENTARY STRUCTURAL ANALYSIS, NORIS AND WILBUR, 2ND EDITION. MCGRAW-HILL BOOK COMPANY, PAGE 304, WORKOUT PROBLEM

UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0; 3 0 35 0; 4 20 0 0; 5 20 20 0; 6 20 35 0; 7 45 0 0; 8 45 20 0; 9 45 35 0; 10 75 0 0; 11 75 20 0; 12 75 35 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 4 5; 4 5 6; 5 7 8; 6 8 9; 7 10 11; 8 11 12; 9 2 5; 10 5 8; 11 8 11; 12 3 6; 13 6 9; 14 9 12;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 10 IZ 240
3 4 PRIS AX 15 IZ 480
5 6 PRIS AX 20 IZ 600
7 8 PRIS AX 12 IZ 360
9 12 PRIS AX 20 IZ 600
10 13 PRIS AX 30 IZ 1200
11 14 PRIS AX 35 IZ 1800
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.32e+06
POISSON 0.290909
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 7 10 FIXED
LOAD 1 HORIZONTAL JOINT LOAD
JOINT LOAD
V. Support Reactions for a Simple Frame

Find support reactions due to a load at the free end of a cantilever bent plate with an intermediate support.

Reference


Problem

Determine the reaction of the system as shown in the figure.
P = 1 kip
L = 10 in

**Figure 377: Cantilever model**

### Comparison

#### Table 395: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>R_x (kips)</td>
<td>1.5</td>
<td>1.5</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\Support Reactions for a Simple Frame.STD is typically installed with the program.

```
STAAD PLANE : SUPPORT REACTIONS FOR A SIMPLE FRAME
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* REFERENCE 'STRENGTH OF MATERIALS' PART-1 BY S. TIMOSHENKO
* PAGE 346 PROBLEM NO. 2. THE ANSWER IN THE BOOK AFTER
* RECALCULATION = 1.5
*
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 0 20 0; 4 10 20 0;
```
STAAD Output

1. STAAD PLANE : SUPPORT REACTIONS FOR A SIMPLE FRAME
INPUT FILE: Support Reactions for a Simple Frame.STD
2. START JOB INFORMATION
3. ENGINEER DATE 17-SEP-18
4. END JOB INFORMATION
5. *
6. * REFERENCE 'STRENGTH OF MATERIALS' PART-1 BY S. TIMOSHENKO
7. * PAGE 346 PROBLEM NO. 2. THE ANSWER IN THE BOOK AFTER
8. * RECALCULATION = 1.5
9. *
10. UNIT INCHES KIP
11. JOINT COORDINATES
12. 1 0 0 0; 2 0 10 0; 3 0 20 0; 4 10 20 0
13. MEMBER INCIDENCES
14. 1 1 2; 2 2 3; 3 3 4
15. MEMBER PROPERTY AMERICAN
16. 1 TO 3 PRIS AX 10 IZ 100
17. DEFINE MATERIAL START
18. ISOTROPIC MATERIAL1
19. E 3000
20. POISSON 0.17
21. END DEFINE MATERIAL
22. CONSTANTS
23. MATERIAL MATERIAL1 ALL
24. SUPPORTS
25. 1 FIXED
26. 2 FIXED BUT FY MZ
27. LOAD 1
28. JOINT LOAD
29. 4 FY -1
30. PERFORM ANALYSIS
31. PRINT SUPPORT REACTION
32. FINISH
20. POISSON 0.17
21. END DEFINE MATERIAL
22. CONSTANTS
23. MATERIAL MATERIAL1 ALL
24. SUPPORTS
25. 1 FIXED
26. 2 FIXED BUT FY MZ
27. LOAD 1
28. JOINT LOAD
29. 4 FY -1.
30. PERFORM ANALYSIS
   : SUPPORT REACTIONS FOR A SIMPLE FRAME -- PAGE NO.
31. PRINT SUPPORT REACTION
   SUPPORT REACTIONS -UNIT KIP INCH STRUCTURE TYPE = PLANE
   -----------------
   JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM Z
   1  1  1.50  1.00  0.00  0.00  0.00  -5.00
   2  1 -1.50  0.00  0.00  0.00  0.00     0.00
   ************** END OF LATEST ANALYSIS RESULT **************
32. FINISH
   *********** END OF THE STAAD.Pro RUN ***********
   **** DATE= APR 14,2019 TIME= 23:1:8 ****
   : SUPPORT REACTIONS FOR A SIMPLE FRAME -- PAGE NO.

V. 2D Portal Reactions 2

Find the maximum moment due to a uniform load on the horizontal member in a 1x1 bay plane frame.
Reference

Problem
Determine the maximum moment in the frame.

E and I same for all members.

\[
L = 20 \text{ ft} \\
w = 2 \text{ kips/ft}
\]

![Figure 378: 1x1 bay plane frame](image)

Comparison

Table 396: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(M_{\text{Max}}) (kip-ft)</td>
<td>44.40</td>
<td>44.44</td>
<td>negligible</td>
</tr>
</tbody>
</table>
STAAD Input

The file \texttt{C:\\Users\\Public\\Documents\\STAAD.Pro CONNECT Edition\\Samples\\Verification Models\\03 Frames\\2D Portal Reactions 2.STD} is typically installed with the program.

STAAD PLANE : 2D PORTAL REACTIONS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* REFERENCE 'STRUCTURAL ANALYSIS' BY JACK C. MCCORMACK,
* PAGE 383 EXAMPLE 22-5, PLANE FRAME WITH NO SIDESWAY
* ANSWER - MAX BENDING = 44.4 FT-KIP
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0; 3 20 20 0; 4 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 1 IZ 0.05
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.132e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 FIXED
LOAD 1
MEMBER LOAD
2 UNI Y -2
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1</td>
<td>1</td>
<td>1</td>
<td>20.00</td>
<td>-3.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-22.21</td>
</tr>
<tr>
<td>2 1</td>
<td>2</td>
<td>2</td>
<td>3.33</td>
<td>20.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-44.44</td>
</tr>
<tr>
<td>3 1</td>
<td>3</td>
<td>3</td>
<td>-3.33</td>
<td>20.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-44.44</td>
</tr>
<tr>
<td>4 -20.00</td>
<td>-3.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>44.44</td>
<td>22.21</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
V. 2D Portal Reactions Sidesway 2

Find the maximum moment due to a concentrated load on the horizontal member in a 1x1 bay plane frame.

Reference

Problem
Determine the maximum moment in the structure.

\[ P = 30 \text{ kip} \]
\[ L_1 = 20 \text{ ft}, \ L_2 = 30 \text{ ft} \]

E and I same for all members

*Figure 379: Unequal leg bay model*
Comparison

Table 397: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_{Max}$ (ft·kip)</td>
<td>69.40</td>
<td>69.44</td>
<td>negligible</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2D Portal Reactions Sidesway 2.STD is typically installed with the program.

STAAD PLANE : 2D PORTAL REACTIONS SIDEWAYS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* PLANE FRAME WITH SIDESWAY. REFERENCE 'STRUCTURAL ANALYSIS'
* BY JACK MCCORMACK. PAGE 385 PROB 22-6.
* ANSWER - MAX BENDING IN MEMB 1 = 69.4 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 10 0; 2 0 30 0; 3 30 30 0; 4 30 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 TABLE ST W12X26
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
END SUPPORTS
1 4 FIXED
LOAD 1 VERTICAL LOAD
MEMBER LOAD
2 CON Y -30 10
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH

STAAD Output

MEMBER END FORCES STRUCTURE TYPE = PLANE
-------------
ALL UNITS ARE -- KIP FEET (LOCAL )
V. 1x2 Plane Frame Lateral Load

Find the maximum moment due to lateral joint loads in a 1x2 bay plane frame.

Reference


Problem

Determine the maximum moment in the frame.

\[
L = 20 \text{ ft}
\]

\[
H_3 = 20 \text{ kip, } H_5 = 10 \text{ kip}
\]

E and I same for all members.

![Figure 380: Two story frame model](image-url)
Comparison

Table 398: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_{Max}$ (ft·kip)</td>
<td>176.40</td>
<td>178.01</td>
<td>0.9%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\1x2 Plane Frame Lateral Load.STD is typically installed with the program.

STAAD PLANE : 1X2 PLANE FRAME LATERAL LOAD
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* MULTIPLE LEVEL PLANE FRAME WITH HORIZONTAL LOAD.
* REFERENCE 'STRUCTURAL ANALYSIS' BY JACK McCORMACK,
* PAGE 388, PROB 22-7. ANSWER - MAX MOM IN MEMB 1 = 176.4 K-F
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0; 3 0 20 0; 4 20 20 0; 5 0 40 0; 6 20 40 0;
MEMBER INCIDENCES
1 1 3; 2 2 4; 3 3 5; 4 4 6; 5 3 4; 6 5 6;
MEMBER PROPERTY AMERICAN
1 TO 6 PRIS AX 0.2 IZ 0.1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 FIXED
LOAD 1 HORIZONTAL LOAD
JOINT LOAD
3 FX 20
5 FX 10
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH

STAAD Output

MEMBER END FORCES STRUCTURE TYPE = PLANE
-----------------
V. 2D Portal Reactions Sidesway 1

To find the displacements at the nodes of a frame due to movements of supports.

Reference

Problem
Calculate the deflections at node B and support D.

*Figure 381: Frame subject to imposed displacements*
Load Cases:
1. Vertical displacement of 1 cm at Node A
2. Vertical displacement of 1 cm at Node B

Comparison

Table 399: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Case 1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Horizontal displacement at node B (cm)</td>
<td>1.25</td>
<td>-1.25</td>
<td>none</td>
</tr>
<tr>
<td>Rotation at node B</td>
<td>0.001667</td>
<td>0.0017</td>
<td>none</td>
</tr>
<tr>
<td>Horizontal displacement at node D (cm)</td>
<td>0.4167</td>
<td>-0.4167</td>
<td>none</td>
</tr>
<tr>
<td>Load Case 2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Horizontal displacement at node B (cm)</td>
<td>1.25</td>
<td>1.25</td>
<td>none</td>
</tr>
<tr>
<td>Rotation at node B</td>
<td>0.001667</td>
<td>0.0017</td>
<td>none</td>
</tr>
<tr>
<td>Horizontal displacement at node D (cm)</td>
<td>0.4167</td>
<td>0.4167</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2D Portal Reactions Sidesway 1.STD is typically installed with the program.

STAAD PLANE DEFLECTIONS DUE TO MOVEMENT OF SUPPORTS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* REFERENCE: INTERMEDIATE STRUCTURAL ANALYSIS, C.K.WANG
* INTERNATIONAL STUDENT EDITION, 1983, MCGRAW HILL
* SECTION 2.11, PAGE 47
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 7.5 0; 3 6 7.5 0; 4 6 2.5 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
UNIT CM KN
MEMBER PROPERTY AMERICAN
V. 2 Bay Frame Moments and Shear

To find the member forces in a 1x2 bay plane frame with members of rectangular section.
Reference


**Problem**

The frame supports a uniformly distributed load and concentrated loads. Calculate the bending moment and shear force at the mid point of the beam of the first bay.

\[ E = 30,000 \text{ ksi} \]

Columns are 12"×24", beams are 12"×30"

![Figure 382: 2 bay frame](image)

**Comparison**

**Table 400: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Case 1</td>
<td>Moment, M (in-kips)</td>
<td>3,375</td>
<td>3,360.15</td>
</tr>
<tr>
<td></td>
<td>Shear, V (kips)</td>
<td>68.75</td>
<td>68.75</td>
</tr>
<tr>
<td>Load Case 2</td>
<td>Moment, M (in-kips)</td>
<td>2,430</td>
<td>2,425.12</td>
</tr>
<tr>
<td></td>
<td>Shear, V (kips)</td>
<td>22.50</td>
<td>22.68</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2 Bay Frame Moments and Shear.STD is typically installed with the program.

STAAD PLANE : 2 BAY FRAME MOMENTS AND SHEAR
START JOB INFORMATION
ENGINEER DATE 09-Oct-17
END JOB INFORMATION

* * * REFERENCE: "MANUAL OF STEEL CONSTRUCTION-ALLOWABLE STRESS DESIGN", AISC, CHICAGO, ILLINOIS, 1989.

UNIT FEET KIP

JOINT COORDINATES
1 0 0 0; 2 18 0 0; 3 36 0 0; 4 0 10 0; 5 18 10 0; 6 36 10 0;

MEMBER INCIDENCES
1 1 4; 2 2 5; 3 3 6; 4 4 5; 5 5 6;

UNIT INCHES KIP

MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 1e+07 IZ 13824
4 5 PRIS AX 1e+07 IZ 27000

MEMBER RELEASE
5 END MZ
4 START MZ

SUPPORTS
1 3 FIXED
2 PINNED

DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.290909
END DEFINE MATERIAL

CONSTANTS

MATERIAL MATERIAL1 ALL
UNIT FEET KIP

LOAD 1 VERTICAL POINT LOADS
JOINT LOAD
4 6 FY -50
5 FY -100

MEMBER LOAD
4 CON GY -100 9
5 CON GY -100 9

LOAD 2 VERTICAL UNIFORM LOADS
MEMBER LOAD
4 UNI GY -10
5 UNI GY -10

PERFORM ANALYSIS

UNIT INCHES KIP

SECTION 0.501 MEMB 4
PRINT MEMBER SECTION FORCES LIST 4
FINISH
STAAD Output

<table>
<thead>
<tr>
<th>MEMB</th>
<th>LOAD</th>
<th>SEC</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>1</td>
<td>0.50</td>
<td>0.00</td>
<td>-68.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-3360.15</td>
</tr>
<tr>
<td>2</td>
<td>0.50</td>
<td>-0.00</td>
<td>-22.68</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2425.12</td>
</tr>
</tbody>
</table>

V. 3D Frame Max Forces

Find the maximum axial force and moment due to load and moment applied at a joint in a Space frame.

Reference


Problem

Determine the maximum axial force and moment in the space structure.

\[ F = 2 \text{ kip}, \ P = 1 \text{ kip}, \ M = 120 \text{ in \cdot kip} \]
\[ L = 120 \text{ in.} \]
\[ E = 30E3 \text{ ksi}, \]
\[ AX = 11 \text{ in}^2 \]
\[ IX = 83 \text{ in}^4 \]
\[ IY = 56 \text{ in}^4 \]
\[ IZ = 56 \text{ in}^4 \]
Comparison

Table 401: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>Advanced Analysis</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_{Max}$ (kips)</td>
<td>1.47</td>
<td>1.47</td>
<td></td>
</tr>
<tr>
<td>$M_{Y, Max}$ (in·kip)</td>
<td>84.04</td>
<td>84.04</td>
<td></td>
</tr>
<tr>
<td>$M_{Z, Max}$ (in·kip)</td>
<td>95.319</td>
<td>95.32</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\3D Frame Max Forces.STD is typically installed with the program.

STAAD SPACE : 3D FRAME MAX FORCES
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION*
* REFERENCE 'COMPUTER PROGRAMS FOR STRUCTURAL ANALYSIS'
* BY WILLIAM WEAVER JR. PAGE 146 STRUCTURE NO. 8.
* ANSWER - MAX AXIAL FORCE= 1.47 (MEMB 3)
* MAX BEND-Y= 84.04, MAX BEND-Z= 95.319 (BOTH MEMB 3)
* UNIT INCHES KIP
JOINT COORDINATES
STAAD Output

### JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.22267</td>
<td>0.00016</td>
<td>-0.17182</td>
<td>-0.00255</td>
<td>0.00217</td>
<td>-0.00213</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.22202</td>
<td>-0.48119</td>
<td>-0.70161</td>
<td>-0.00802</td>
<td>0.00101</td>
<td>-0.00435</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>

### SUPPORT REACTIONS -UNIT KIP INCH  STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1</td>
<td>-1.10</td>
<td>-0.43</td>
<td>0.22</td>
<td>48.78</td>
<td>-17.97</td>
<td>96.12</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-0.90</td>
<td>1.43</td>
<td>-0.22</td>
<td>123.08</td>
<td>47.25</td>
<td>-11.72</td>
</tr>
</tbody>
</table>

### MEMBER END FORCES  STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.90</td>
<td>-0.43</td>
<td>0.22</td>
<td>22.71</td>
<td>-17.97</td>
<td>-36.37</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>3</td>
<td>-0.90</td>
<td>0.43</td>
<td>-0.22</td>
<td>-22.71</td>
<td>-34.18</td>
<td>-67.36</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>3</td>
<td>-0.43</td>
<td>1.10</td>
<td>0.22</td>
<td>-17.97</td>
<td>-48.78</td>
<td>96.12</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>2</td>
<td>0.43</td>
<td>-1.10</td>
<td>-0.22</td>
<td>17.97</td>
<td>22.71</td>
<td>36.37</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>2</td>
<td>1.47</td>
<td>-0.71</td>
<td>-0.48</td>
<td>-37.02</td>
<td>15.69</td>
<td>-53.28</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-1.47</td>
<td>0.71</td>
<td>0.48</td>
<td>37.02</td>
<td>84.04</td>
<td>-95.32</td>
<td></td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

34. FINISH
V.04 Plate and Shell Elements

V. Cantilever Tube Stresses and Deflection

To find deflections and element stresses due to loads at the free end of a Cantilever beam of tubular section. The beam is modeled using plate/shell elements.

Reference

Problem
A cantilever beam is made of a tubular section. Using plate/shell elements calculate the deflection at the free end and axial stress at the center of the beam for the following free end loads:

\[
\begin{align*}
P &= 1,000 \text{ lb} \\
M_x &= 2,000 \text{ in} \cdot \text{lb} \\
M_y &= 2,500 \text{ in} \cdot \text{lb} \\
V &= 1,000 \text{ lb}
\end{align*}
\]
Figure 384: Cantilever beam modeled with elements
Figure 385: STAAD.Pro Model showing Node numbers
Figure 386: Element numbers at a) the middle section and b) the free end
Average deflection for nodes 11, 22, 33, 44, 55, 66, 77, 88, 99 and 10 due to load case 3 ($M_y$):
\[ d = \frac{4(0.07431) + 4(0.07384) + 2(0.07374)}{10} = 0.074008 \]

Average deflection for nodes 11, 22, 33, 44, 55, 66, 77, 88, 99 and 10 due to load case 4 ($V$):
\[ d = \frac{4(-0.41123) + 4(-0.41124) + 2(-0.41057)}{10} = 0.41110 \]

Average bending stress for nodes 76, 86, 126, and 136 due to load case 3 ($M_y$):
\[ \sigma = 4(5551.52)/4 = 5551.52 \text{ psi} \]

Average bending stress for nodes 71, 81, 121, and 131 due to load case 4 ($V$):
\[ \sigma = 4(42074.63)/4 = 42074.63 \text{ psi} \]

**Comparison**

**Table 402: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free end deflection due to axial load, $P$ (in)</td>
<td>Nodes 11, 44, 66, 99</td>
<td>0.004</td>
<td>0.00410</td>
</tr>
<tr>
<td></td>
<td>Nodes 22, 33, 55, 77, 88, 110</td>
<td>0.004</td>
<td>0.00401</td>
</tr>
<tr>
<td>Axial stress at the middle of the beam due to axial load, $P$ (psi)</td>
<td>Elements 46, 66, 96, 116</td>
<td>2,000</td>
<td>2,000.02</td>
</tr>
<tr>
<td></td>
<td>Elements 56, 106</td>
<td>2,000</td>
<td>1,991.57</td>
</tr>
<tr>
<td></td>
<td>Elements 76, 86, 126, 136</td>
<td>2,000</td>
<td>2,004,19</td>
</tr>
<tr>
<td>In plane shear stress at the free end due to torque, $M_x$ (psi)</td>
<td>Elements 50, 70, 100, 120</td>
<td>3,333</td>
<td>3,333.55</td>
</tr>
<tr>
<td></td>
<td>Elements 60, 110</td>
<td>3,333</td>
<td>3,325.54</td>
</tr>
<tr>
<td></td>
<td>Elements 80, 90, 130, 140</td>
<td>3,333</td>
<td>3,342.23</td>
</tr>
<tr>
<td>Avg. free end deflection at the center due to moment, $M_y$ (in)</td>
<td>0.0741</td>
<td>0.074008</td>
<td>negligible</td>
</tr>
<tr>
<td>Avg. free end bending stress at the center due to moment, $M_y$ (psi)</td>
<td>5,647</td>
<td>5,551.52</td>
<td>1.7%</td>
</tr>
<tr>
<td>Avg. free end deflection at the center due to shear, $V$ (in)</td>
<td>0.4152</td>
<td>-0.41110</td>
<td>1.0%</td>
</tr>
<tr>
<td>Avg. free end bending stress at the center due to shear, $V$ (psi)</td>
<td>42,913</td>
<td>42,074.63</td>
<td>2.0%</td>
</tr>
</tbody>
</table>
Verification Examples
V.04 Plate and Shell Elements

STAAD Input
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Cantilever Tube Stresses and Deflection.STD is typically installed with the program.

STAAD SPACE: A CANTILEVER BEAM OF TUBULAR SECTION
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* REFERENCES: 1. TIMOSHENKO, S., STRENGTH OF MATERIALS, PART 1,
2ND EDITION, D. VAN NOSTRAND COMPANY, 1940
2. SEELEY, F. B., AND SMITH, J.O., ADVANCED MECHANICS
OF MATERIALS, 2ND EDITION, JOHN WILEY AND SONS, 1955

UNIT INCHES POUND

JOINT COORDINATES
1 0 0 0; 2 2 0 0; 3 4 0 0; 4 6 0 0; 5 8 0 0; 6 10 0 0; 7 12 0 0;
8 14 0 0; 9 16 0 0; 10 18 0 0; 12 0 1 0; 13 2 1 0; 14 4 1 0;
15 6 1 0; 16 8 1 0; 17 10 1 0; 19 14 1 0; 20 16 1 0;
21 18 1 0; 22 20 1 0; 23 0 2 0; 24 2 2 0; 25 4 2 0; 26 6 2 0; 27 8 2 0;
28 10 2 0; 29 12 2 0; 30 14 2 0; 32 18 2 0; 33 20 2 0;
34 0 3 0; 35 2 3 0; 36 4 3 0; 37 6 3 0; 38 8 3 0; 39 10 3 0; 40 12 3 0;
41 14 3 0; 42 16 3 0; 43 18 3 0; 44 20 3 0; 45 0 3 1; 46 2 3 1;
47 4 3 1; 48 6 3 1; 49 8 3 1; 50 10 3 1; 51 12 3 1; 52 14 3 1;
53 16 3 1; 54 18 3 1; 55 20 3 1; 56 0 3 2; 57 2 3 2; 58 4 3 2; 59 6 3 2;
60 8 3 2; 61 10 3 2; 62 12 3 2; 63 14 3 2; 64 16 3 2; 65 18 3 2;
66 20 3 2; 67 0 2 2; 68 2 2 2; 69 4 2 2; 70 6 2 2; 71 8 2 2; 72 10 2 2;
73 12 2 2; 74 14 2 2; 75 16 2 2; 76 18 2 2; 77 20 2 2; 78 0 1 2;
79 2 1 2; 80 4 1 2; 81 6 1 2; 82 8 1 2; 83 10 1 2; 84 12 1 2; 85 14 1 2;
86 16 1 2; 87 18 1 2; 88 20 1 2; 89 0 0 2; 90 2 0 2; 91 4 0 2; 92 6 0 2;
93 8 0 2; 94 10 0 2; 95 12 0 2; 96 14 0 2; 97 16 0 2; 98 18 0 2;
99 20 0 2; 100 0 0 1; 101 2 0 1; 102 4 0 1; 103 6 0 1; 104 8 0 1;
105 10 0 1; 106 12 0 1; 107 14 0 1; 108 16 0 1; 109 18 0 1; 110 20 0 1;

MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 34 35; 12 35 36; 13 36 37; 14 37 38; 15 38 39; 16 39 40;
17 40 41; 18 41 42; 19 42 43; 20 43 44; 21 56 57; 22 57 58; 23 58 59;
24 59 60; 25 60 61; 26 61 62; 27 62 63; 28 63 64; 29 64 65; 30 65 66;
31 89 90; 32 90 91; 33 91 92; 34 92 93; 35 93 94; 36 94 95; 37 95 96;
38 96 97; 39 97 98; 40 98 99;

ELEMENT INCIDENCES SHELL
41 12 13 14; 42 2 3 13 14; 43 3 4 15 14; 44 4 5 16 15; 45 5 6 17 16;
46 7 18 17; 47 7 19 18; 48 8 20 19; 49 9 21 20; 50 10 21 21;
51 12 13 24 23; 52 13 14 25 24; 53 14 15 26 25; 54 15 16 27 26;
55 16 17 28 27; 56 17 18 29 28; 57 18 19 30 29; 58 19 20 31 30;
59 20 21 32 31; 60 21 22 33 32; 61 23 24 35 34; 62 24 25 36 35;
63 25 26 37 36; 64 26 27 38 37; 65 27 28 39 38; 66 28 29 40 39;
67 29 30 41 40; 68 30 31 42 41; 69 31 32 43 42; 70 32 33 44 43;
71 34 35 46 45; 72 35 36 47 46; 73 36 37 48 47; 74 37 38 49 48;
75 38 39 50 49; 76 39 40 51 50; 77 40 41 52 51; 78 41 42 53 52;
79 42 43 54 53; 80 43 44 55 54; 81 45 46 57 56; 82 46 47 58 57;
83 47 48 59 58; 84 48 49 60 59; 85 49 50 61 60; 86 50 51 62 61;
87 51 52 63 62; 88 52 53 64 63; 89 53 54 65 64; 90 54 55 66 65;
91 56 57 68 67; 92 57 58 69 68; 93 58 59 70 69; 94 59 60 71 70;
95 60 61 72 71; 96 61 62 73 72; 97 62 63 74 73; 98 63 64 75 74;
Verification Examples
V.04 Plate and Shell Elements

99 64 65 76 75; 100 65 66 77 76; 101 67 68 79 78; 102 68 69 80 79;
103 73 74 85 84; 104 74 75 86 85; 105 75 76 87 86; 110 76 77 88 87;
111 78 79 90 89; 112 79 80 91 90; 113 80 81 92 91; 114 81 82 93 92;
115 82 83 94 93; 116 83 84 95 94; 117 84 85 96 95; 118 85 86 97 96;
119 86 87 98 97; 120 87 88 99 98; 121 88 90 101 100; 122 90 91 102 101;
123 91 92 103 102; 124 92 93 104 103; 125 93 94 105 104;
126 94 95 106 105; 127 95 96 107 106; 128 96 97 108 107;
129 97 98 109 108; 130 98 99 110 109; 131 100 101 2 1; 132 101 102 3 2;
133 102 103 4 3; 134 103 104 5 4; 135 104 105 6 5; 136 105 106 7 6;
137 106 107 8 7; 138 107 108 9 8; 139 108 109 10 9; 140 109 110 11 10;
MEMBER TRUSS
1 TO 40
MEMBER PROPERTY AMERICAN
1 TO 40 PRIS AX 0.0005
ELEMENT PROPERTY
41 TO 140 THICKNESS 0.05
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1000
POISSON 0.599866
ISOTROPIC MATERIAL2
E 1e+07
POISSON 0.33
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 MEMB 1 TO 40
MATERIAL MATERIAL2 MEMB 41 TO 140
SUPPORTS
1 12 23 34 45 56 67 78 89 100 FIXED
LOAD 1 AXIAL LOAD
JOINT LOAD
11 22 33 44 55 66 77 88 99 110 FX 100
LOAD 2 TORQUE
JOINT LOAD
11 FY 80.43 FZ -86.21
22 FY 169.44
44 FY 80.43 FZ 86.21
55 FZ 162.23
66 FY -80.43 FZ 86.21
77 FY -169.44
88 FY -80.43 FZ -86.21
110 FZ -162.23
LOAD 3 END MOMENT
JOINT LOAD
44 55 66 FX -258.6
33 77 FX -86.3
22 88 FX 86.3
11 99 110 FX 258.6
LOAD 4 SHEAR
JOINT LOAD
11 44 66 99 FY -66
22 33 77 88 FY -184
PERFORM ANALYSIS
LOAD LIST 1 TO 4
PRINT JOINT DISPLACEMENTS LIST 11 22 33 44 55 66 77 88 99 110
LOAD LIST 1
PRINT ELEMENT STRESSES LIST 46 56 66 76 86 96 106 116 126 136
### STAAD Output

**JOINT DISPLACEMENT (INCH RADIANS)**  
**STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>1</td>
<td>0.00410</td>
<td>0.00024</td>
<td>0.00020</td>
<td>-0.00005</td>
<td>-0.00082</td>
<td>0.00093</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>-0.00035</td>
<td>0.01443</td>
<td>-0.02268</td>
<td>0.01498</td>
<td>0.00067</td>
<td>0.00001</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0.01142</td>
<td>0.07431</td>
<td>0.0056</td>
<td>0.0039</td>
<td>-0.0233</td>
<td>0.00846</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>-0.04451</td>
<td>-0.41123</td>
<td>-0.0026</td>
<td>-0.0119</td>
<td>0.00184</td>
<td>-0.02863</td>
</tr>
<tr>
<td>22</td>
<td>1</td>
<td>0.00401</td>
<td>0.00004</td>
<td>-0.0001</td>
<td>-0.0010</td>
<td>0.00013</td>
<td>0.00012</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>-0.00011</td>
<td>0.01446</td>
<td>-0.0757</td>
<td>0.01505</td>
<td>0.00058</td>
<td>0.00007</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0.00371</td>
<td>0.07384</td>
<td>0.0015</td>
<td>-0.0031</td>
<td>0.00034</td>
<td>0.00638</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>-0.01476</td>
<td>-0.41124</td>
<td>-0.0036</td>
<td>0.0050</td>
<td>-0.0013</td>
<td>-0.02764</td>
</tr>
<tr>
<td>33</td>
<td>1</td>
<td>0.00401</td>
<td>-0.00004</td>
<td>-0.0001</td>
<td>0.0010</td>
<td>0.00013</td>
<td>0.00012</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.00011</td>
<td>0.01446</td>
<td>0.0757</td>
<td>0.01505</td>
<td>0.00058</td>
<td>0.00007</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-0.00371</td>
<td>0.07384</td>
<td>-0.0015</td>
<td>-0.0031</td>
<td>-0.0034</td>
<td>0.00638</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.01476</td>
<td>-0.41124</td>
<td>0.0036</td>
<td>0.0050</td>
<td>0.0013</td>
<td>0.02764</td>
</tr>
<tr>
<td>44</td>
<td>1</td>
<td>0.00410</td>
<td>-0.00024</td>
<td>0.00020</td>
<td>-0.0005</td>
<td>-0.00082</td>
<td>0.00093</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.00035</td>
<td>0.01443</td>
<td>0.02268</td>
<td>0.01498</td>
<td>-0.00067</td>
<td>0.00001</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-0.01142</td>
<td>0.07431</td>
<td>-0.0056</td>
<td>0.0039</td>
<td>0.00233</td>
<td>0.00846</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.04451</td>
<td>-0.41123</td>
<td>0.0026</td>
<td>0.00119</td>
<td>-0.00184</td>
<td>-0.02863</td>
</tr>
<tr>
<td>55</td>
<td>1</td>
<td>0.00401</td>
<td>-0.00011</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00027</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.02265</td>
<td>0.01404</td>
<td>-0.00059</td>
<td>0.00000</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-0.01111</td>
<td>0.07374</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00696</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.04414</td>
<td>-0.41057</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.03079</td>
</tr>
<tr>
<td>66</td>
<td>1</td>
<td>0.00410</td>
<td>-0.00024</td>
<td>-0.00020</td>
<td>-0.0005</td>
<td>0.00082</td>
<td>-0.00093</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>-0.00035</td>
<td>-0.01443</td>
<td>0.02268</td>
<td>0.01498</td>
<td>0.00067</td>
<td>-0.00001</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-0.01142</td>
<td>0.07431</td>
<td>0.0056</td>
<td>-0.0039</td>
<td>0.00233</td>
<td>0.00846</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.04451</td>
<td>-0.41123</td>
<td>0.0026</td>
<td>0.00119</td>
<td>0.00184</td>
<td>0.02863</td>
</tr>
<tr>
<td>77</td>
<td>1</td>
<td>0.00401</td>
<td>0.00004</td>
<td>0.00001</td>
<td>-0.00010</td>
<td>-0.00013</td>
<td>0.00012</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>-0.00011</td>
<td>-0.01446</td>
<td>0.0757</td>
<td>0.01505</td>
<td>-0.00058</td>
<td>-0.00007</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-0.00371</td>
<td>0.07384</td>
<td>0.0015</td>
<td>0.0031</td>
<td>0.0034</td>
<td>0.00638</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.01476</td>
<td>-0.41124</td>
<td>0.0036</td>
<td>0.0050</td>
<td>0.0013</td>
<td>-0.02764</td>
</tr>
<tr>
<td>88</td>
<td>1</td>
<td>0.00401</td>
<td>0.00004</td>
<td>0.00001</td>
<td>0.00010</td>
<td>-0.0013</td>
<td>0.00012</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.00011</td>
<td>-0.01446</td>
<td>0.0757</td>
<td>0.01505</td>
<td>0.00058</td>
<td>0.00007</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0.00371</td>
<td>0.07384</td>
<td>0.0015</td>
<td>0.0031</td>
<td>0.0034</td>
<td>0.00638</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>-0.01476</td>
<td>-0.41124</td>
<td>0.0036</td>
<td>-0.0050</td>
<td>0.0013</td>
<td>-0.02764</td>
</tr>
<tr>
<td>99</td>
<td>1</td>
<td>0.00410</td>
<td>0.00024</td>
<td>0.00020</td>
<td>0.0005</td>
<td>0.00082</td>
<td>0.00093</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.00035</td>
<td>-0.01443</td>
<td>0.02268</td>
<td>0.01498</td>
<td>0.00067</td>
<td>-0.00001</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-0.01142</td>
<td>0.07431</td>
<td>-0.0056</td>
<td>-0.0039</td>
<td>0.00233</td>
<td>0.00846</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.04451</td>
<td>-0.41123</td>
<td>0.0026</td>
<td>0.00119</td>
<td>-0.00184</td>
<td>-0.02863</td>
</tr>
<tr>
<td>110</td>
<td>1</td>
<td>0.00401</td>
<td>0.00011</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.0027</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.02265</td>
<td>0.01404</td>
<td>0.00059</td>
<td>0.00000</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0.01111</td>
<td>0.07374</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00696</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>-0.04414</td>
<td>-0.41057</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.03079</td>
</tr>
</tbody>
</table>

---

**A CANTILEVER BEAM OF TUBULAR SECTION**  
**-- PAGE NO.**
**ELEMENT STRESSES**  FORCE, LENGTH UNITS = POUND INCH

<table>
<thead>
<tr>
<th>Element</th>
<th>Load</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
<th>SX</th>
<th>SY</th>
<th>SXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>46</td>
<td>1</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>2000.02</td>
<td>2.49</td>
<td>-0.24</td>
</tr>
<tr>
<td>1998.93</td>
<td>1998.62</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2000.58</td>
<td>1999.46</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 2000.58</td>
<td>SMIN= 3.01</td>
<td>TMAX= 998.04</td>
<td>ANGLE= -0.1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1999.46</td>
<td>SMIN= 1.67</td>
<td>TMAX= 998.89</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>56</td>
<td>1</td>
<td>-0.01</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>1994.35</td>
<td>1994.87</td>
<td>1991.57</td>
</tr>
<tr>
<td>TOP : SMAX= 1991.55</td>
<td>SMIN= -5.60</td>
<td>TMAX= 998.57</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1991.59</td>
<td>SMIN= -6.53</td>
<td>TMAX= 999.06</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>66</td>
<td>1</td>
<td>0.02</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2000.58</td>
<td>1998.62</td>
<td>2000.02</td>
</tr>
<tr>
<td>2003.44</td>
<td>2003.71</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2004.95</td>
<td>1998.44</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 2000.58</td>
<td>SMIN= 3.01</td>
<td>TMAX= 998.04</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1999.46</td>
<td>SMIN= 1.67</td>
<td>TMAX= 998.89</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>76</td>
<td>1</td>
<td>-0.04</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>2003.32</td>
<td>2003.71</td>
<td>2004.19</td>
</tr>
<tr>
<td>2003.44</td>
<td>2004.95</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 2003.44</td>
<td>SMIN= 0.20</td>
<td>TMAX= 1001.60</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 2004.95</td>
<td>SMIN= 2.48</td>
<td>TMAX= 1001.24</td>
<td>ANGLE= 0.1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>86</td>
<td>1</td>
<td>-0.04</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>2003.32</td>
<td>2003.71</td>
<td>2004.19</td>
</tr>
<tr>
<td>2003.44</td>
<td>2004.95</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 2003.44</td>
<td>SMIN= 0.20</td>
<td>TMAX= 1001.60</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 2004.95</td>
<td>SMIN= 2.48</td>
<td>TMAX= 1001.24</td>
<td>ANGLE= 0.1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>96</td>
<td>1</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2000.58</td>
<td>1999.46</td>
<td></td>
</tr>
<tr>
<td>2000.02</td>
<td>2.49</td>
<td>-0.24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 2000.58</td>
<td>SMIN= 3.01</td>
<td>TMAX= 998.04</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1999.46</td>
<td>SMIN= 1.67</td>
<td>TMAX= 998.89</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>106</td>
<td>1</td>
<td>-0.01</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>1994.35</td>
<td>1994.87</td>
<td>1991.57</td>
</tr>
<tr>
<td>TOP : SMAX= 1991.55</td>
<td>SMIN= -5.60</td>
<td>TMAX= 998.57</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1991.59</td>
<td>SMIN= -6.53</td>
<td>TMAX= 999.06</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>116</td>
<td>1</td>
<td>0.02</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>1998.93</td>
<td>1998.62</td>
<td>2000.02</td>
</tr>
<tr>
<td>2000.58</td>
<td>1999.46</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 2000.58</td>
<td>SMIN= 3.01</td>
<td>TMAX= 998.04</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1999.46</td>
<td>SMIN= 1.67</td>
<td>TMAX= 998.89</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.04 Plate and Shell Elements

<table>
<thead>
<tr>
<th>PLATE NO.</th>
<th>76</th>
<th>56</th>
<th>76</th>
<th>76</th>
<th>76</th>
</tr>
</thead>
<tbody>
<tr>
<td>CASE NO.</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

**MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

<table>
<thead>
<tr>
<th>MAXIMUM PRINCIPAL STRESS</th>
<th>MINIMUM PRINCIPAL STRESS</th>
<th>MAXIMUM SHEAR STRESS</th>
<th>MAXIMUM VONMISES STRESS</th>
<th>MAXIMUM TRESCA STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.004949E+03</td>
<td>-6.534180E+00</td>
<td>1.001600E+03</td>
<td>2.003712E+03</td>
<td>2.004949E+03</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PLATE NO.</th>
<th>76</th>
<th>56</th>
<th>76</th>
<th>76</th>
<th>76</th>
</tr>
</thead>
<tbody>
<tr>
<td>CASE NO.</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

**ELEMENT STRESSES**

**FORCE, LENGTH UNITS = POUN INCH**

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>-2.88</td>
<td>1.41</td>
<td>-0.01</td>
<td>-0.02</td>
<td>-0.03</td>
</tr>
<tr>
<td></td>
<td>5636.28</td>
<td>5911.95</td>
<td>-0.36</td>
<td>18.30</td>
<td>3333.55</td>
</tr>
<tr>
<td></td>
<td>6508.14</td>
<td>6826.34</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| 60           | 0.00 | 1.42 | 0.00 | -0.00 | -0.08 |
|              | 5445.08 | 6074.93 |      |      |      |
|              | 6287.44 | 7014.73 |      |      |      |

| 70           | 2.88 | 1.41 | 0.01 | 0.02 | -0.03 |
|              | 5636.28 | 5911.95 | 0.36 | -18.30 | 3333.55 |
|              | 6508.14 | 6826.34 |      |      |      |

| 80           | -3.78 | 1.42 | 0.02 | 0.06 | -0.03 |
|              | 5674.31 | 5906.56 | 1.32 | 11.09 | 3342.23 |
|              | 6551.26 | 6819.29 |      |      |      |

| 90           | -2.88 | 1.41 | 0.02 | 0.06 | -0.03 |
|              | 5674.31 | 5906.56 | 1.32 | 11.09 | 3342.23 |
|              | 6551.26 | 6819.29 |      |      |      |

**A CANTILEVER BEAM OF TUBULAR SECTION**

---

**LOAD LIST 2**

**PRINT ELEMENT STRESSES LIST 50 60 80 90 100 110 120 130 140**

---

**USER MANUAL**
**Verification Examples**

**V.04 Plate and Shell Elements**

<table>
<thead>
<tr>
<th>TOP</th>
<th>SMAX</th>
<th>SMIN</th>
<th>TMAX</th>
<th>ANGLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>110</td>
<td>2</td>
<td>0.00</td>
<td>1.42</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5445.08</td>
<td>6074.93</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>BOTT</th>
<th>SMAX</th>
<th>SMIN</th>
<th>TMAX</th>
<th>ANGLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>110</td>
<td>2</td>
<td>2.88</td>
<td>1.41</td>
<td>0.01</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5636.28</td>
<td>5911.95</td>
<td>0.36</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TOP</th>
<th>SMAX</th>
<th>SMIN</th>
<th>TMAX</th>
<th>ANGLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>120</td>
<td>2</td>
<td>3.78</td>
<td>-1.42</td>
<td>0.02</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5674.31</td>
<td>5906.56</td>
<td>1.32</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>BOTT</th>
<th>SMAX</th>
<th>SMIN</th>
<th>TMAX</th>
<th>ANGLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>120</td>
<td>2</td>
<td>3.78</td>
<td>-1.42</td>
<td>-0.02</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5674.31</td>
<td>5906.56</td>
<td>-1.32</td>
</tr>
</tbody>
</table>

### A CANTILEVER BEAM OF TUBULAR SECTION

<table>
<thead>
<tr>
<th>10 ELEMENT STRESSES</th>
<th>FORCE, LENGTH UNITS = POUN INCH</th>
</tr>
</thead>
</table>

**STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH**

**ELEMENT LOAD**

<table>
<thead>
<tr>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
</table>

**VONT | VONB | SX | SY | SXY |

**TRESCAT | TRESCLAB**

| 130  | 2 | -3.78 | -1.42 | 0.02  |
|      |   | 5674.31 | 5906.56 | 1.32 | -11.09 |

| 140  | 2 | 3.78 | -1.42 | -0.02 |
|      |   | 5674.31 | 5906.56 | -1.32 | 11.09 |

### MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES

<table>
<thead>
<tr>
<th>MAXIMUM PRINCIPAL STRESS</th>
<th>MINIMUM PRINCIPAL STRESS</th>
<th>SHEAR STRESS</th>
<th>VONMISES STRESS</th>
<th>TRESCA STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.511688E+03</td>
<td>-3.511688E+03</td>
<td>3.507364E+03</td>
<td>6.074933E+03</td>
<td>7.014728E+03</td>
</tr>
</tbody>
</table>

### END OF ELEMENT FORCES

**11 LOAD LIST 3**

**113 PRINT ELEMENT STRESSES LIST 45 46 55 56 65 66 75 76 85 86 95 96 105**

**114. 106 115 116 125 126 135 136**

### A CANTILEVER BEAM OF TUBULAR SECTION

<table>
<thead>
<tr>
<th>11 ELEMENT STRESSES</th>
<th>FORCE, LENGTH UNITS = POUN INCH</th>
</tr>
</thead>
</table>

**STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH**

**ELEMENT LOAD**

<table>
<thead>
<tr>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
</table>

**VONT | VONB | SX | SY | SXY |

**TRESCAT | TRESCLAB**

| 45   | 3 | -0.02 | 0.01 | -0.00 | -0.00 |
|      |   | 3703.79 | 3702.75 | 1.06 | -0.17 |

| 46   | 3 | 0.03 | 0.03 | -0.00 | -0.00 |
|      |   | 3704.99 | 3703.08 | 10.35 | -1.86 |

---

**STAAD.Pro 3305 User Manual**
### Verification Examples
#### V.04 Plate and Shell Elements

A CANTILEVER BEAM OF TUBULAR SECTION

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
<th>TRESCAT</th>
<th>TRESCAB</th>
</tr>
</thead>
<tbody>
<tr>
<td>85</td>
<td>3</td>
<td>-0.81</td>
<td>0.01</td>
<td>0.04</td>
<td>-0.00</td>
<td>-0.01</td>
<td>5463.48</td>
<td>5646.25</td>
</tr>
<tr>
<td>86</td>
<td>3</td>
<td>-1.04</td>
<td>0.03</td>
<td>0.04</td>
<td>-0.00</td>
<td>-0.01</td>
<td>5459.11</td>
<td>5640.90</td>
</tr>
<tr>
<td>95</td>
<td>3</td>
<td>0.02</td>
<td>-0.01</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>3703.79</td>
<td>3702.75</td>
</tr>
<tr>
<td>96</td>
<td>3</td>
<td>-0.03</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>3704.99</td>
<td>3703.08</td>
</tr>
</tbody>
</table>

**Force, Length Units: Pounch**

**Stress = Force/Unit Width/Thick, Moment = Force-Length/Unit Width**

---

STAAD.Pro 3306 User Manual
### Element Stresses

**FORCES, LENGTH UNITS = POUND INCH**

**STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT</td>
<td>VONB</td>
<td>SX</td>
<td>SY</td>
<td>SXY</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TRESCAT</td>
<td>TRESCAB</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table of Data**

<table>
<thead>
<tr>
<th>PLATE NO.</th>
<th>125</th>
<th>75</th>
<th>75</th>
<th>75</th>
<th>75</th>
<th>75</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOP : SMAX</td>
<td>-8.31</td>
<td>-3709.14</td>
<td>-3709.27</td>
<td>1850.41</td>
<td>ANGLE</td>
<td>90.0</td>
</tr>
<tr>
<td>BOTT: SMAX</td>
<td>-12.39</td>
<td>-3709.14</td>
<td>-3709.27</td>
<td>1848.44</td>
<td>ANGLE</td>
<td>90.0</td>
</tr>
<tr>
<td>105</td>
<td>3</td>
<td>-0.00</td>
<td>-0.02</td>
<td>-0.00</td>
<td>-0.00</td>
<td>1.29</td>
</tr>
<tr>
<td>3.32</td>
<td>1.83</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX</td>
<td>1.66</td>
<td>-1.66</td>
<td>1.66</td>
<td>ANGLE</td>
<td>90.0</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX</td>
<td>0.92</td>
<td>-0.92</td>
<td>0.92</td>
<td>ANGLE</td>
<td>90.0</td>
<td></td>
</tr>
<tr>
<td>106</td>
<td>3</td>
<td>-0.00</td>
<td>-0.03</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-1.65</td>
</tr>
<tr>
<td>4.19</td>
<td>1.52</td>
<td>0.00</td>
<td>0.00</td>
<td>1.29</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.32</td>
<td>1.83</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX</td>
<td>1.66</td>
<td>-1.66</td>
<td>1.66</td>
<td>ANGLE</td>
<td>90.0</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX</td>
<td>0.92</td>
<td>-0.92</td>
<td>0.92</td>
<td>ANGLE</td>
<td>90.0</td>
<td></td>
</tr>
<tr>
<td>115</td>
<td>3</td>
<td>-0.02</td>
<td>-0.01</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>3703.79</td>
<td>3702.75</td>
<td>3703.80</td>
<td>1.06</td>
<td>0.17</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3704.34</td>
<td>3704.36</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX</td>
<td>5.645007E+03</td>
<td>-5.645007E+03</td>
<td>2.823742E+03</td>
<td>5.646246E+03</td>
<td>5.647484E+03</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX</td>
<td>5.645007E+03</td>
<td>-5.645007E+03</td>
<td>2.823742E+03</td>
<td>5.646246E+03</td>
<td>5.647484E+03</td>
<td></td>
</tr>
<tr>
<td>126</td>
<td>3</td>
<td>-0.03</td>
<td>-0.03</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3704.99</td>
<td>3703.08</td>
<td>3709.20</td>
<td>10.35</td>
<td>1.86</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3709.14</td>
<td>3709.27</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX</td>
<td>5.645007E+03</td>
<td>-5.645007E+03</td>
<td>2.823742E+03</td>
<td>5.646246E+03</td>
<td>5.647484E+03</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX</td>
<td>5.645007E+03</td>
<td>-5.645007E+03</td>
<td>2.823742E+03</td>
<td>5.646246E+03</td>
<td>5.647484E+03</td>
<td></td>
</tr>
<tr>
<td>135</td>
<td>3</td>
<td>-0.03</td>
<td>-0.03</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3704.99</td>
<td>3703.08</td>
<td>3709.20</td>
<td>10.35</td>
<td>1.86</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3709.14</td>
<td>3709.27</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX</td>
<td>5.645007E+03</td>
<td>-5.645007E+03</td>
<td>2.823742E+03</td>
<td>5.646246E+03</td>
<td>5.647484E+03</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX</td>
<td>5.645007E+03</td>
<td>-5.645007E+03</td>
<td>2.823742E+03</td>
<td>5.646246E+03</td>
<td>5.647484E+03</td>
<td></td>
</tr>
<tr>
<td>136</td>
<td>3</td>
<td>-0.03</td>
<td>-0.03</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5459.11</td>
<td>5640.90</td>
<td>5551.52</td>
<td>3.75</td>
<td>4.91</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Verification Examples**

V.04 Plate and Shell Elements

**STAAD.Pro User Manual**
### Verification Examples

**V.04 Plate and Shell Elements**

---

**STAAD.Pro User Manual**

---

115. LOAD LIST 4

116. PRINT ELEMENT STRESSES LIST 41 51 61 71 81 91 101 111 121 131

**ELEMENT STRESSES LIST 41**

:A CANTILEVER BEAM OF TUBULAR SECTION

---

**ELEMENT STRESSES**

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>41</td>
<td>-14.20</td>
<td>10.26</td>
<td>-0.04</td>
<td>-0.03</td>
<td>0.01</td>
</tr>
<tr>
<td></td>
<td>28183.86</td>
<td>28061.26</td>
<td>-28424.34</td>
<td>-2007.57</td>
<td>-3462.32</td>
</tr>
<tr>
<td></td>
<td>28963.74</td>
<td>28777.50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>-1630.57</td>
<td>SMIN=</td>
<td>-28963.74</td>
<td>TMAX=</td>
<td>13666.59</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>-1492.00</td>
<td>SMIN=</td>
<td>-28777.50</td>
<td>TMAX=</td>
<td>13642.75</td>
</tr>
<tr>
<td>51</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>-0.00</td>
<td>-1.14</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.02</td>
</tr>
<tr>
<td></td>
<td>5356.82</td>
<td>5159.93</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>6185.52</td>
<td>5947.78</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>3092.76</td>
<td>SMIN=</td>
<td>-3092.76</td>
<td>TMAX=</td>
<td>3092.76</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>2973.89</td>
<td>SMIN=</td>
<td>-2973.89</td>
<td>TMAX=</td>
<td>2973.89</td>
</tr>
<tr>
<td>61</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>14.20</td>
<td>10.26</td>
<td>0.04</td>
<td>0.03</td>
<td>0.01</td>
</tr>
<tr>
<td></td>
<td>28183.86</td>
<td>28061.26</td>
<td>28424.34</td>
<td>2007.57</td>
<td>-3462.32</td>
</tr>
<tr>
<td></td>
<td>28963.74</td>
<td>28777.50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>28963.74</td>
<td>SMIN=</td>
<td>1630.57</td>
<td>TMAX=</td>
<td>13666.59</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>28777.50</td>
<td>SMIN=</td>
<td>1492.00</td>
<td>TMAX=</td>
<td>13642.75</td>
</tr>
<tr>
<td>71</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>21.02</td>
<td>-14.62</td>
<td>-0.33</td>
<td>-0.06</td>
<td>-0.06</td>
</tr>
<tr>
<td></td>
<td>39671.99</td>
<td>41111.66</td>
<td>42074.63</td>
<td>3650.80</td>
<td>692.96</td>
</tr>
<tr>
<td></td>
<td>41300.99</td>
<td>42874.24</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>41300.99</td>
<td>SMIN=</td>
<td>3488.38</td>
<td>TMAX=</td>
<td>18906.30</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>42874.24</td>
<td>SMIN=</td>
<td>3787.25</td>
<td>TMAX=</td>
<td>19543.49</td>
</tr>
<tr>
<td>81</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>21.02</td>
<td>14.62</td>
<td>-0.33</td>
<td>-0.06</td>
<td>0.06</td>
</tr>
<tr>
<td></td>
<td>39671.99</td>
<td>41111.66</td>
<td>42074.63</td>
<td>3650.80</td>
<td>-692.96</td>
</tr>
<tr>
<td></td>
<td>41300.99</td>
<td>42874.24</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>41300.99</td>
<td>SMIN=</td>
<td>3488.38</td>
<td>TMAX=</td>
<td>18906.30</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>42874.24</td>
<td>SMIN=</td>
<td>3787.25</td>
<td>TMAX=</td>
<td>19543.49</td>
</tr>
<tr>
<td>91</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>14.20</td>
<td>10.26</td>
<td>0.04</td>
<td>0.03</td>
<td>-0.01</td>
</tr>
<tr>
<td></td>
<td>28183.86</td>
<td>28061.26</td>
<td>28424.34</td>
<td>2007.57</td>
<td>3462.32</td>
</tr>
<tr>
<td></td>
<td>28963.74</td>
<td>28777.50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>28963.74</td>
<td>SMIN=</td>
<td>1630.57</td>
<td>TMAX=</td>
<td>13666.59</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>28777.50</td>
<td>SMIN=</td>
<td>1492.00</td>
<td>TMAX=</td>
<td>13642.75</td>
</tr>
<tr>
<td>101</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.00</td>
<td>1.14</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.02</td>
</tr>
<tr>
<td></td>
<td>5356.82</td>
<td>5159.93</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>6185.52</td>
<td>5947.78</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>3092.76</td>
<td>SMIN=</td>
<td>-3092.76</td>
<td>TMAX=</td>
<td>3092.76</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>2973.89</td>
<td>SMIN=</td>
<td>-2973.89</td>
<td>TMAX=</td>
<td>2973.89</td>
</tr>
<tr>
<td>111</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>-14.20</td>
<td>-18.26</td>
<td>-0.04</td>
<td>-0.03</td>
<td>-0.01</td>
</tr>
<tr>
<td></td>
<td>28183.86</td>
<td>28061.26</td>
<td>-28424.34</td>
<td>2007.57</td>
<td>3462.32</td>
</tr>
<tr>
<td></td>
<td>28963.74</td>
<td>28777.50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP: SMAX=</td>
<td>-1630.57</td>
<td>SMIN=</td>
<td>-28963.74</td>
<td>TMAX=</td>
<td>13666.59</td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>-1492.00</td>
<td>SMIN=</td>
<td>-28777.50</td>
<td>TMAX=</td>
<td>13642.75</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro User Manual**

---

"***************END OF ELEMENT FORCES***************"
### V. 2D Cantilever Beam End Deflection 1

To find the free end deflection due to a joint load on a Cantilever beam modeled using Plate/shell elements.

#### Reference

Hand calculation.

#### Problem

Using the finite element method calculate the deflection of the free end of the cantilever beam.

#### Figure 387: Fixed support beam with temperature load

![Figure 387](image)

**E** = 4,278 ksi

**h** = 10 in

---

**Verification Examples**

**V.04 Plate and Shell Elements**

<table>
<thead>
<tr>
<th>VONT</th>
<th>VONB</th>
<th>SX</th>
<th>SY</th>
<th>SXY</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TRESCAT</th>
<th>TRESCAB</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>121</th>
<th>4</th>
<th>-21.02</th>
<th>14.62</th>
<th>0.33</th>
<th>0.06</th>
<th>0.06</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>39671.99</td>
<td>41111.66</td>
<td>-42074.63</td>
<td>-3650.80</td>
<td>-692.96</td>
</tr>
<tr>
<td></td>
<td></td>
<td>41300.99</td>
<td>42874.24</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**TOP : SMAX= -3488.38 SMIN= -41300.99 TMAX= 18906.30 ANGLE= -89.2**

**BOTT: SMAX= -3787.25 SMIN= -42874.24 TMAX= 19543.49 ANGLE= -88.8**

<table>
<thead>
<tr>
<th>131</th>
<th>4</th>
<th>-21.02</th>
<th>-14.62</th>
<th>0.33</th>
<th>0.06</th>
<th>-0.06</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>39671.99</td>
<td>41111.66</td>
<td>-42074.63</td>
<td>-3650.80</td>
<td>692.96</td>
</tr>
<tr>
<td></td>
<td></td>
<td>41300.99</td>
<td>42874.24</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**TOP : SMAX= -3488.38 SMIN= -41300.99 TMAX= 18906.30 ANGLE= 89.2**

**BOTT: SMAX= -3787.25 SMIN= -42874.24 TMAX= 19543.49 ANGLE= 88.8**

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

<table>
<thead>
<tr>
<th>PLATE NO.</th>
<th>CASE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>71</td>
<td>4</td>
</tr>
<tr>
<td>121</td>
<td>4</td>
</tr>
<tr>
<td>71</td>
<td>4</td>
</tr>
<tr>
<td>71</td>
<td>4</td>
</tr>
<tr>
<td>71</td>
<td>4</td>
</tr>
</tbody>
</table>

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

<table>
<thead>
<tr>
<th>MAXIMUM</th>
<th>MINIMUM</th>
<th>MAXIMUM</th>
<th>MAXIMUM</th>
<th>MAXIMUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRINCIPAL</td>
<td>PRINCIPAL</td>
<td>SHEAR</td>
<td>VONMISES</td>
<td>TRESCA</td>
</tr>
<tr>
<td>STRESS</td>
<td>STRESS</td>
<td>STRESS</td>
<td>STRESS</td>
<td>STRESS</td>
</tr>
<tr>
<td>4.287424E+04</td>
<td>-4.287424E+04</td>
<td>1.954349E+04</td>
<td>4.111166E+04</td>
<td>4.287424E+04</td>
</tr>
</tbody>
</table>

********************END OF ELEMENT FORCES********************
Theoretical Solution

Moment of Inertia:

\[ I = \frac{(5 \text{ in})(10 \text{ in})^3}{12} = 416.7 \text{ in}^4 \]

Deflection at free end:

\[ \delta = \frac{PL^3}{3EI} = \frac{2 \text{ kips}(60 \text{ in})^3}{3(4,278 \text{ ksi})(416.7 \text{ in}^4)} = 0.0808 \text{ in} \]

Comparison

Table 403: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection at node B</td>
<td>0.0808</td>
<td>0.08245</td>
<td>2.1%</td>
<td></td>
</tr>
</tbody>
</table>

The expression for deflection for beam is used to compare the results of a model with Plate Elements with FE formulation, hence the difference in results.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Cantilever Beam End Deflection 1.STD is typically installed with the program.

STAAD SPACE : A CANTILEVER BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

*REFERENCE : HAND CALCULATION

UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 2 0; 3 0 4 0; 4 0 6 0; 5 0 8 0; 6 0 10 0; 7 2.5 0 0;
8 2.5 2 0; 9 2.5 4 0; 10 2.5 6 0; 11 2.5 8 0; 12 2.5 10 0; 13 5 0 0;
14 5 2 0; 15 5 4 0; 16 5 6 0; 17 5 8 0; 18 5 10 0; 19 7.5 0 0;
20 7.5 2 0; 21 7.5 4 0; 22 7.5 6 0; 23 7.5 8 0; 24 7.5 10 0; 25 10 0 0;
26 10 2 0; 27 10 4 0; 28 10 6 0; 29 10 8 0; 30 10 10 0; 31 12.5 0 0;
32 12.5 2 0; 33 12.5 4 0; 34 12.5 6 0; 35 12.5 8 0; 36 12.5 10 0;
Verification Examples

V.04 Plate and Shell Elements

<table>
<thead>
<tr>
<th>ELEMENT INCIDENCES SHELL</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 7 1 2 8; 2 8 2 3 9; 3 9 3 4 10; 4 10 4 5 11; 5 11 5 6 12; 6 13 7 8 14;</td>
</tr>
<tr>
<td>7 14 8 9 15; 8 15 9 10 16; 9 16 10 11 17; 10 17 11 12 18;</td>
</tr>
<tr>
<td>11 19 13 14 20; 12 20 14 15 21; 13 21 15 16 22; 14 22 16 17 23;</td>
</tr>
<tr>
<td>15 23 17 18 24; 16 24 17 20 26; 17 26 20 21 27; 18 27 21 22 28;</td>
</tr>
<tr>
<td>19 28 22 23 29; 20 29 23 24 30; 21 31 25 26 32; 22 32 26 27 33;</td>
</tr>
<tr>
<td>23 33 27 28 34; 24 34 28 29 35; 25 35 29 30 36; 26 37 31 32 38;</td>
</tr>
<tr>
<td>27 38 32 33 39; 28 39 33 34 40; 29 40 34 35 41; 30 41 35 36 42;</td>
</tr>
<tr>
<td>31 43 37 38 44; 32 44 38 39 45; 33 45 39 40 46; 34 46 40 41 47;</td>
</tr>
<tr>
<td>35 47 41 42 48; 36 49 43 44 50; 37 50 44 45 51; 38 51 45 46 52;</td>
</tr>
<tr>
<td>39 52 46 47 53; 40 53 47 48 54; 41 55 49 50 56; 42 56 50 51 57;</td>
</tr>
<tr>
<td>43 57 51 52 58; 44 58 52 53 59; 45 59 53 54 60; 46 61 55 56 62;</td>
</tr>
<tr>
<td>47 62 56 57 63; 48 63 57 58 64; 49 64 58 59 65; 50 65 59 60 66;</td>
</tr>
<tr>
<td>51 66 60 61 62; 52 68 62 63 69; 53 69 63 64 70; 54 70 64 65 71;</td>
</tr>
<tr>
<td>55 71 65 66 72; 56 73 67 68 74; 57 74 68 69 75; 58 75 69 70 76;</td>
</tr>
<tr>
<td>59 76 70 71 77; 60 77 71 72 78; 61 79 73 74 80; 62 80 74 75 81;</td>
</tr>
<tr>
<td>63 81 75 76 82; 64 82 76 77 83; 65 83 77 78 84; 66 85 79 80 86;</td>
</tr>
<tr>
<td>67 86 80 81 87; 68 87 81 82 88; 69 88 82 83 89; 70 89 83 84 90;</td>
</tr>
<tr>
<td>71 91 85 86 92; 72 92 86 87 93; 73 93 87 88 94; 74 94 88 89 95;</td>
</tr>
<tr>
<td>75 95 89 90 96; 76 97 91 92 98; 77 98 92 93 99; 78 99 93 94 100;</td>
</tr>
<tr>
<td>79 100 94 95 101; 80 101 95 96 102; 81 102 97 98 104; 82 104 98 99 105;</td>
</tr>
<tr>
<td>83 105 99 100 106; 84 106 100 101 107; 85 107 101 102 108;</td>
</tr>
<tr>
<td>86 109 103 104 110; 87 110 104 105 111; 88 111 105 106 112;</td>
</tr>
<tr>
<td>89 112 106 107 113; 90 113 107 108 114; 91 115 109 110 116;</td>
</tr>
<tr>
<td>92 116 110 111 117; 93 117 111 112 118; 94 118 112 113 119;</td>
</tr>
<tr>
<td>95 119 113 114 120; 96 121 115 116 122; 97 122 116 117 123;</td>
</tr>
<tr>
<td>98 123 117 118 124; 99 124 118 119 125; 100 125 119 120 126;</td>
</tr>
<tr>
<td>101 127 121 122 128; 102 128 122 123 129; 103 129 123 124 130;</td>
</tr>
<tr>
<td>104 130 124 125 131; 105 131 125 126 132; 106 132 127 128 134;</td>
</tr>
<tr>
<td>107 134 128 129 135; 108 135 129 130 136; 109 136 130 131 137;</td>
</tr>
<tr>
<td>110 137 131 132 138; 111 139 133 134 140; 112 140 134 135 141;</td>
</tr>
<tr>
<td>113 141 135 136 142; 114 142 136 137 143; 115 143 137 138 144;</td>
</tr>
<tr>
<td>116 145 139 140 146; 117 146 140 141 147; 118 147 141 142 148;</td>
</tr>
<tr>
<td>119 148 142 143 149; 120 149 143 144 150;</td>
</tr>
</tbody>
</table>

ELEMENT PROPERTY
1 TO 120 THICKNESS 5
DEF MATERIAL START

STAAD.Pro 3311 User Manual
**V. 2D Cantilever Beam End Deflection 2**

Find the deflection and moments for plate-bending finite element due to a pressure load.

**Reference**

Simple hand calculation by considering the entire structure as a cantilever beam.

**Problem**

A simple cantilever plate is divided into 12 4-noded finite elements. A uniform pressure load is applied and the maximum deflection at the tip of the cantilever and the maximum bending at the support are calculated.

Plate thickness = 25 mm
Uniform pressure = 5 N/mm²
Plate length = 6 spaces at 50 mm = 300 mm
Plate width = 2 space at 50 mm = 100 mm
Figure 388: Finite element mesh of cantilevered plate.

Theoretical Solution

Maximum deflection is equal to \( \frac{WL^2}{8EI} \), where:

\[
\Delta_{\text{max}} = \frac{5(300)(100)(300)^3}{8(210 \cdot 10^3)(100 \cdot 25^3)} = \frac{4050(10)^9}{218.75(10)^9} = 18.51\text{mm}
\]

Maximum Moment:

\[
M_{\text{max}} = \frac{WL}{2} = \frac{5(300)(100)(300)}{2} = 22.5(10)^6\text{N}\cdot\text{mm}
\]

Comparison

Table 404: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \delta_{\text{max}} ) (mm)</td>
<td>18.51</td>
<td>18.159</td>
<td>1.9%</td>
</tr>
<tr>
<td>( M_{\text{max}} ) (kN\cdot m)</td>
<td>22.50</td>
<td>22.50</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: The maximum moment is taken as the sum of the moments at nodes 1, 8, and 15 (i.e., \( 5.47 + 11.56 + 5.47 = 22.5 \text{ kN}\cdot\text{m} \)).

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Cantilever Beam End Deflection 2.STD is typically installed with the program.

STAAD SPACE : DEFORMATION OF A CANTILEVER PLATE UNDER UNIFORM PRESSURE
START JOB INFORMATION
* DEFORMATION OF A CANTILEVER PLATE UNDER UNIFORM PRESSURE.
* COMPARISON WITH ESTABLISHED FORMULA (WL^3/8EI)
*
UNIT MMS KN
JOINT COORDINATES
1 0 0 0; 2 50 0 0; 3 100 0 0; 4 150 0 0; 5 200 0 0; 6 250 0 0;
7 300 0 0; 8 0 50 0; 9 50 50 0; 10 100 50 0; 11 150 50 0; 12 200 50 0;
13 250 50 0; 14 300 50 0; 15 0 100 0; 16 50 100 0; 17 100 100 0;
18 150 100 0; 19 200 100 0; 20 250 100 0; 21 300 100 0;

ELEMENT INCIDENCES SHELL
1 1 2 9 8; 2 2 3 10 9; 3 3 4 11 10; 4 4 5 12 11; 5 5 6 13 12;
6 6 7 14 13; 7 8 9 16 15; 8 9 10 17 16; 9 10 11 18 17; 10 11 12 19 18;
11 12 13 20 19; 12 13 14 21 20;

ELEMENT PROPERTY
1 TO 12 THICKNESS 25
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.1e+08
POISSON 0.3
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 8 15 FIXED
UNIT MMS NEWTON
LOAD 1 5N/SQ.MM. UNIFORM LOAD
ELEMENT LOAD
1 TO 12 PR 5
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 14
UNIT METER KN
PRINT SUPPORT REACTION
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>1.8159</td>
<td>0.0000</td>
<td>-0.0813</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

38. UNIT METER KN
39. PRINT SUPPORT REACTION

: DEFORMATION OF A CANTILEVER PLATE UNDER UNIFORM PRESSURE -- PAGE NO. 4

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-18.91</td>
<td>-1.54</td>
<td>5.47</td>
<td>0.00</td>
</tr>
</tbody>
</table>
V. 2D Curved Beam Maximum Stress

Using plate/shell elements, find maximum bending stress due to a force couple on a curved cantilever beam.

Reference

Problem
Find the maximum bending stress

\[ E = 3,000.0 \text{ ksi.} \]

Poisson’s ratio = 0.3.

\[ t = 1.0 \text{ in.} \]

\[ P = 100 \text{ lbs} \]
Figure 389: Cantilevered, curved plate with coupling load at free end

**Comparison**

Table 405: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inside stress (psi)</td>
<td>655.0</td>
<td>636.02</td>
<td>2.9%</td>
<td>The result from the Beam theory is compared with the result from a Finite Element Model output, hence the difference in results.</td>
</tr>
<tr>
<td>Result Type</td>
<td>Theory</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>------------------------------</td>
<td>--------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>Outside stress (psi)</td>
<td>555.0</td>
<td>556.73</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Curved Beam Maximum Stress.STD is typically installed with the program.

```plaintext
STAAD SPACE: CURVED BEAM WITH PLATE ELEMENTS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE: TIMOSHENKO, S., "STRENGTH OF MATERIALS, PART I, ELEMENTARY
*       THEORY AND PROBLEMS", 3RD EDITION, D. VAN NOSTRAND CO.,
*       INC., NEW YORK, 1956.
*
UNIT INCHES POUND
JOINT COORDINATES
1 4.5 0 0; 2 0 4.5 0; 3 0.781 4.432 0; 4 1.539 4.299 0; 5 2.25 3.897 0;
6 2.893 3.447 0; 7 3.447 2.893 0; 8 3.897 2.25 0; 9 4.229 1.539 0;
10 4.432 0.781 0; 11 4.25 0 0; 12 0 4.25 0; 13 0.738 4.185 0;
14 1.454 3.994 0; 15 2.125 3.681 0; 16 2.732 3.256 0; 17 3.256 2.732 0;
18 3.681 2.125 0; 19 3.994 1.454 0; 20 4.185 0.738 0; 21 4 0 0;
22 0 4 0; 23 0.695 3.939 0; 24 1.368 3.759 0; 25 2 3.464 0;
26 2.571 3.064 0; 27 3.064 2.571 0; 28 3.464 2 0; 29 3.759 1.368 0;
30 3.939 0.695 0; 31 3.75 0 0; 32 0 3.75 0; 33 0.651 3.693 0;
34 1.283 3.524 0; 35 1.875 3.248 0; 36 2.41 2.873 0; 37 2.873 2.41 0;
38 3.248 1.875 0; 39 3.524 1.283 0; 40 3.693 0.651 0; 41 3.5 0 0;
42 0 3.5 0; 43 0.608 3.447 0; 44 1.197 3.289 0; 45 1.75 3.031 0;
46 2.25 2.681 0; 47 2.681 2.25 0; 48 3.031 1.75 0; 49 3.289 1.197 0;
50 3.447 0.608 0;
ELEMENT INCIDENCES SHELL
1  2  3 13 12; 2  3  4 14 13; 3  4  5 15 14; 4  5  6 16 15; 5  6  7 17 16;
6  7  8 18 17; 7  8  9 19 18; 8  9 10 20 19; 9 10  1 11 20; 10 12 13 23 22;
11 13 14 24 23; 12 14 15 25 24; 13 15 16 26 25; 14 16 17 27 26;
15 17 18 28 27; 16 18 19 29 28; 17 19 20 30 29; 18 20 11 21 30;
19 22 23 33 32; 20 23 24 34 33; 21 24 25 35 34; 22 25 26 36 35;
23 26 27 37 36; 24 27 28 38 37; 25 28 29 39 38; 26 29 30 40 39;
27 30 31 41 40; 28 32 33 42 41; 29 33 34 43 42; 30 34 35 44 43;
31 35 36 45 44; 32 36 37 46 45; 33 37 38 48 47; 34 38 39 49 48;
35 39 40 50 49; 36 40 31 41 50;
ELEMENT PROPERTY
1 TO 36 THICKNESS 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
```
SUPPORTS
1 11 21 31 41 FIXED
LOAD 1 100 IN-IB
JOINT LOAD
2 FX 100
42 FX -100
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT ELEMENT JOINT STRESSES LIST 9 36
FINISH

STAAD Output

SUPPORT REACTIONS -UNIT POUN INCH STRUCTURE TYPE = SPACE
------------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z   MOM-X   MOM-Y   MOM Z
1  1    -15.30     61.91      0.00      0.00      0.00    -2.09
11 1     3.80      69.53      0.00      0.00      0.00     1.21
21 1    12.47      5.36      0.00      0.00      0.00     1.75
31 1    11.58     -66.95      0.00      0.00      0.00
41 1    -12.55    -69.85      0.00      0.00      0.00

************** END OF LATEST ANALYSIS RESULT **************

52. PRINT ELEMENT JOINT STRESSES LIST 9 36
ELEMENT  JOINT STRESSES
:CURVED BEAM WITH PLATE ELEMENTS

4

ELEMENT STRESSES

FORCE,LENGTH UNITS= POUN INCH
------------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

ELEMENT  LOAD   SQX    SQY    MX     MY     MXY
VONT   VONB   SX     SY     SXY
TRESCAT TRESCAB
9  1  432.15  432.15  -433.11   -2.49    8.94
433.30  433.30
TOP : SMAX= -2.31 SMIN= -433.30 TMAX= 215.50 ANGLE= 88.8
BOTT: SMAX= -2.31 SMIN= -433.30 TMAX= 215.50 ANGLE= 88.8
JOINT 0.00 0.00 0.00 0.00 0.00
10  592.15  592.15  -560.35   58.12    15.37
527.90  527.90
TOP : SMAX= 58.50 SMIN= -560.73 TMAX= 309.62 ANGLE= 88.6
BOTT: SMAX= 58.50 SMIN= -560.73 TMAX= 309.62 ANGLE= 88.6
JOINT 0.00 0.00 0.00 0.00 0.00
1   527.90  527.90  -556.47   -63.65    11.40
527.90  527.90
TOP : SMAX= -63.39 SMIN= -556.73 TMAX= 246.67 ANGLE= 88.7
BOTT: SMAX= -63.39 SMIN= -556.73 TMAX= 246.67 ANGLE= 88.7
JOINT 0.00 0.00 0.00 0.00 0.00
11  285.05  285.05  -310.91   -64.85    13.47
285.05  285.05
TOP : SMAX= -64.11 SMIN= -311.65 TMAX= 123.77 ANGLE= 86.9
BOTT: SMAX= -64.11 SMIN= -311.65 TMAX= 123.77 ANGLE= 86.9
JOINT 0.00 0.00 0.00 0.00 0.00
20  351.99  351.99  -318.48   -88.91    8.48
351.99  351.99
TOP : SMAX= 59.15 SMIN= -318.67 TMAX= 188.91 ANGLE= 88.7
BOTT: SMAX= 59.15 SMIN= -318.67 TMAX= 188.91 ANGLE= 88.7
JOINT 0.00 0.00 0.00 0.00 0.00
36  1  457.10  457.10  473.43   34.66     2.24
473.44  473.44
V. 2D Triangular Surface with Thermal Load

A simply-supported equilateral triangle is subjected to a linear thermal gradient. The deflections and bending moments are calculated.

Reference


Problem

The purpose of this example is to demonstrate the ability of STAAD to calculate the response of a structure when thermally-loaded plate elements are utilized. The STAAD.Pro model considers one-half of the plate, modeled using 49 quad elements and seven tri plate elements. For specifying inclined supports, an auxiliary coordinate system is defined along the edge (see second figure below) not parallel to a global axis.

\[ E = 10^{6} \text{ lb/in}^2 \]
\[ v = 0.3 \]
\[ \alpha = 12 \times 10^{-6} \text{ in/in } \text{oF} \]
\[ \Delta T = T_{\text{top}} - T_{\text{bottom}} = 4500 \]
\[ t = 0.1 \text{ inches} \]
\[ a = 3.0 \text{ inches} \]
Figure 390: Simply-supported, equilateral triangle with thermal load
**Theoretical Solution**

From page 96 of the reference:

\[
M_x = - \frac{a A E t^2}{24} \left( 1 + \frac{3x'}{a} \right)
\]

\[
= - \frac{12(10)^{-6}(450.0)10(10)^6(0.1)^2}{24} \left( 1 + \frac{3x'}{3} \right) = - 22.5 \left( 1 + x' \right)
\]

\[
M_y = - \frac{a A E t^2}{24} \left( 1 - \frac{3x'}{a} \right) = - 22.5 \left( 1 - x' \right)
\]

\[
M_{xy} = \frac{a A E t^2 y}{8a}
\]

\[
= \frac{12(10)^{-6}(450.0)10(10)^6(0.1)^2}{8(3)} y = 22.5 y
\]

\[
w = \frac{a A T(1 + v)}{4at} \left( x'^3 - 3y^2 x' - ax'^2 - ay^2 + \frac{4a^3}{27} \right)
\]

\[
= 0.00585 \left( x'^3 - 3y^2 x' - 3x'^2 - 3y^2 + 4 \right)
\]

The origin of the x' axis in the reference is located at a/3, or 1 inch from the edge. Therefore:

\[x' = x - 1\]
Upon substitution:
\[
\begin{align*}
M_x &= -22.5x \\
M_y &= -22.5(2 - x) \\
M_{xy} &= 22.5y \\
w &= 0.00585(x^3 - 6x^2 + 9x - 3y^2x)
\end{align*}
\]

Comparison

Table 406: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z deflection at center of element along x-axis (in)</td>
<td>Node 6</td>
<td>0.02331</td>
<td>0.02331</td>
</tr>
<tr>
<td>Maximum (magnitude) bending moment along x-axis (in-lb/in)</td>
<td>Mx at Element 8</td>
<td>-36.16</td>
<td>-36.15</td>
</tr>
<tr>
<td></td>
<td>Mx at Element 14</td>
<td>-64.29</td>
<td>-65.64</td>
</tr>
<tr>
<td></td>
<td>My at Element 1</td>
<td>-42.59</td>
<td>-42.59</td>
</tr>
<tr>
<td></td>
<td>My at Element 8</td>
<td>-8.839</td>
<td>-8.838</td>
</tr>
<tr>
<td>Twisting moment along y-axis at center of elements (in-lb/in)</td>
<td>Mxy at Element 55</td>
<td>36.19</td>
<td>36.06</td>
</tr>
</tbody>
</table>

The results show excellent agreement for displacements and for quad plate element stresses except in the few instances where the magnitude of the answers is very small. The tri plate elements are lower order elements, and it is therefore not unusual for these elements to show poorer agreement than quads where mesh size is smaller.

STAAD Input

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Triangular Surface with Thermal Load.STD is typically installed with the program.

```
STAAD SPACE :SIMPLY SUPPORTED PLATE THERMAL
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* AN EQUILATERAL TRIANGLE WITH LINEAR THERMAL GRADIENT
* 
* EXPECTED ANSWERS
```
**Verification Examples**

**V.04 Plate and Shell Elements**

```
* DEFLECTION OF NODE 6, IN THE X3 DIRECTION
  THEORY .02331
  STAAD .02331
*
* INTERNAL MOMENTS IN QUAD PLATE NO 8
  THEORY  -36.16  -8.864
  STAAD  -36.15  -8.84
*
*
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1  0  0  0;  2  0.214286  0  0;  3  0.428571  0  0;  4  0.642857  0  0;  5  0.857143  0  0;
  6  1.07143  0  0;  7  1.28571  0  0;  8  1.5  0  0;  9  1.71429  0  0; 10  1.92857  0  0;
 11  2.14286  0  0; 12  2.35714  0  0; 13  2.57143  0  0; 14  2.78571  0  0; 15  3  0  0;
 16  2.78571  0.123718  0; 17  0  0.247436  0; 18  0.214286  0.247436  0;
 19  0.428571  0.247436  0; 20  0.642857  0.247436  0; 21  0.857143  0.247436  0;
 22  1.07143  0.247436  0; 23  1.28571  0.247436  0; 24  1.5  0.247436  0;
 25  1.71429  0.247436  0; 26  1.92857  0.247436  0; 27  2.14286  0.247436  0;
 28  2.35714  0.247436  0; 29  2.57143  0.247436  0; 30  2.35714  0.371154  0;
 31  0  0.494872  0; 32  0.214286  0.494872  0; 33  0.428571  0.494872  0;
 34  0.642857  0.494872  0; 35  0.857143  0.494872  0; 36  1.07143  0.494872  0;
 37  1.28571  0.494872  0; 38  1.5  0.494872  0; 39  1.71429  0.494872  0;
 40  1.92857  0.494872  0; 41  2.14286  0.494872  0; 42  2.35714  0.494872  0;
 43  0.742307  0; 44  0.214286  0.742307  0; 45  0.428571  0.742307  0;
 46  0.642857  0.742307  0; 47  0.857143  0.742307  0; 48  1.07143  0.742307  0;
 49  1.28571  0.742307  0; 50  1.5  0.742307  0; 51  1.71429  0.742307  0;
 52  1.5  0.866025  0; 53  0  0.989743  0; 54  0.214286  0.989743  0;
 55  0.428571  0.989743  0; 56  0.642857  0.989743  0; 57  0.857143  0.989743  0;
 58  1.07143  0.989743  0; 59  1.28571  0.989743  0; 60  1.5  0.989743  0;
 61  1.71429  0.989743  0; 62  2.14286  0.989743  0; 63  2.35714  0.989743  0;
 64  2.57143  0.989743  0; 65  0.742307  0; 66  0.214286  0.742307  0;
 67  0.428571  0.742307  0; 68  0.642857  0.742307  0; 69  0.857143  0.742307  0;
 70  1.07143  0.742307  0; 71  1.28571  0.742307  0;
*70 3 1.732050
ELEMENT INCIDENCES SHELL
  1  1  2  18  17;  2  2  3  19  18;  3  3  4  20  19;  4  4  5  21  20;  5  5  6  22  21;  6  6  7  23  22;
  7  7  8  24  23;  8  8  9  25  24;  9  9 10  26  25; 10 10 11  27  26; 11 11 12  28  27;
 12 12 13  29  28; 13 13 14  30  29; 14 14 15  31  30; 15 15 16  32  31; 16 16 17  33  32;
 17 17 18  34  33; 18 18 20  35  34; 19 19 21  36  35; 20 20 22  37  36; 21 21 23  38  37;
 22 22 24  39  38; 23 23 25  40  39; 24 24 26  41  40; 25 25 27  42  41; 26 26 28  43  42;
 27 27 31  44  43; 28 28 32  45  44; 29 29 33  46  47; 30 30 34  47  48; 31 31 35  48  47;
 32 32 36  49  48; 33 33 37  50  49; 34 34 38  51  50; 35 35 39  52  51; 36 36 40  53  52;
 37 37 41  54  53; 38 38 42  55  54; 39 39 43  56  55; 40 40 44  57  56; 41 41 45  58  57;
 42 42 46  59  58; 43 43 47  60  59; 44 44 48  61  60; 45 45 49  62  61; 46 46 50  63  62;
 47 47 51  64  63; 48 48 52  65  64; 49 49 53  66  65; 50 50 54  67  66; 51 51 55  68  67;
 52 52 56  69  68; 53 53 57  70  69; 54 54 58  71  70; 55 55 60  72  71; 56 56 61  73  72;
DEFINE MATERIAL START
ISOTROPIC ALUMINUM
  E  1e+07
  POISSON  0.3
  ALPHA  1.2e-05
END DEFINE MATERIAL
CONSTANTS
MATERIAL ALUMINUM ALL
```
ELEMENT PROPERTY
1 TO 56 THICKNESS 0.1
SUPPORTS
16 29 30 41 52 59 60 65 66 69 70 INC REFJT 15 FIXED BUT FX MX MZ
2 TO 14 FIXED BUT FX FZ MY MZ
17 31 43 53 61 67 FIXED BUT FY MY MZ
1 FIXED BUT MY MZ
15 71 FIXED
LOAD 1 UNIFORM BENDING TEMPERATURE
TEMPERATURE LOAD
1 TO 56 TEMP 0 450
PERFORM ANALYSIS PRINT STATICS CHECK
PRINT JOINT DISPLACEMENTS LIST 2 TO 14
PRINT ELEMENT JOINT STRESSES LIST 1 8 14 55
FINISH

STAAD Output

JOINT       DISPLACE LIST     2
:SIMPLY SUPPORTED PLATE THERMAL                          -- PAGE NO.

5
JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
-------------------------------------
JOINT LOAD X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN
--
2 1 0.00000 0.00000 0.00973 0.00000 -0.03853 0.00000
3 1 0.00000 0.00000 0.01658 0.00000 -0.02590 0.00000
4 1 0.00000 0.00000 0.02090 0.00000 -0.01489 0.00000
5 1 0.00000 0.00000 0.02303 0.00000 -0.00549 0.00000
6 1 0.00000 0.00000 0.02331 0.00000 0.00230 0.00000
7 1 0.00000 0.00000 0.02211 0.00000 0.00848 0.00000
8 1 0.00000 0.00000 0.01975 0.00000 0.01305 0.00000
9 1 0.00000 0.00000 0.01658 0.00000 0.01600 0.00000
10 1 0.00000 0.00000 0.01295 0.00000 0.01735 0.00000
11 1 0.00000 0.00000 0.00921 0.00000 0.01709 0.00000
12 1 0.00000 0.00000 0.00570 0.00000 0.01518 0.00000
13 1 0.00000 0.00000 0.00276 0.00000 0.01165 0.00000
14 1 0.00000 0.00000 0.00075 0.00000 0.00672 0.00000

************** END OF LATEST ANALYSIS RESULT **************

77. PRINT ELEMENT JOINT STRESSES LIST 1 8 14 55
ELEMENT JOINT STRESSES LIST
:SIMPLY SUPPORTED PLATE THERMAL                          -- PAGE NO.

6
ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
-------------------------------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
---
ELEMENT LOAD SQX SQY MX MY MXY
VONT VONB SX SY SXY
TRESCAT TRESPAC
1 1 -0.01 -0.01 -2.41 -42.59 2.78
25029.83 25029.83 0.00 0.00 0.00
25668.88 25668.88

TOP : SMAX= -1331.23 SMIN= -25668.88 TMAX= 12168.82 ANGLE= 3.9
BOTT: SMAX= 25668.88 SMIN= 1331.23 TMAX= 12168.82 ANGLE= -86.1
JINT -0.00 -0.01 -1.67 -45.50 1.93
1 26888.39 26888.39 0.00 0.00 0.00
TOP : SMAX= -951.54 SMIN= -27351.53 TMAX= 13200.00 ANGLE= 2.5

Verification Examples
V.04 Plate and Shell Elements

STAAD.Pro
3324
User Manual
<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>STRESS</th>
<th>VONB</th>
<th>MXY</th>
<th>VONT</th>
<th>TRESCAT</th>
<th>TRESCAB</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT  1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>45910.85</td>
<td>45910.85</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10032.26</td>
<td>50421.84</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT  14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>45571.45</td>
<td>45571.45</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10886.75</td>
<td>50421.84</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT  17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>46298.03</td>
<td>46298.03</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10886.64</td>
<td>50421.84</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT  20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>45571.45</td>
<td>45571.45</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10886.75</td>
<td>50421.84</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Verification Examples**

V.04 Plate and Shell Elements

STAAD.Pro 3325 User Manual
V. 2D Circular Surface Displacements and Stresses

A circular plate is fixed along its perimeter. Using plate/shell elements, find the deflection at the center, maximum bending stress due to a uniformly distributed load, and a concentrated load at the center.

Reference


Problem

The circular plate shown below is subject to two load cases. Load 1 is a uniform pressure, \( w \), and load 2 is a concentrated force, \( P \), at the center. Determine:

- deflection at the center for both load cases
- bending stress at the support for both load cases
- moment at the center for load case 1

\[
E = 30,000.0 \text{ ksi} \\
\text{Poisson’s ratio} = 0.3 \\
r = 40 \text{ in.} \\
t = 1 \text{ in.}
\]
\[ w = 6 \text{ psi.} \]
\[ P = 7,539.82 \text{ lbs} \]

**Comparison**

**Table 407: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Case 1</td>
<td>( \sigma_{\text{bend}} ) (psi)</td>
<td>7,200</td>
<td>6,887(^{a})</td>
<td>4.3%</td>
</tr>
<tr>
<td></td>
<td>( \delta_{\text{max}} ) (in) (Y translation at Node 127)</td>
<td>-0.0874</td>
<td>-0.08618</td>
<td>1.4%</td>
</tr>
<tr>
<td></td>
<td>Moment at center (in·lb/in)</td>
<td>780</td>
<td>806.1(^{b})</td>
<td>3.4%</td>
</tr>
<tr>
<td>Load Case 2</td>
<td>( \sigma_{\text{bend}} ) (psi)</td>
<td>3,600</td>
<td>3,700(^{c})</td>
<td>2.8%</td>
</tr>
<tr>
<td></td>
<td>( \delta_{\text{max}} ) (in) (Y translation at Node 127)</td>
<td>-0.0874</td>
<td>-0.08646</td>
<td>1.0%</td>
</tr>
</tbody>
</table>

\(^{a}\) In element 9 at node 10, \( M_y = 1,147.91 \text{ in·lb/in} \). This yields \( \sigma_{\text{bend}} = 1,147.91(6) = 6,887 \text{ psi} \).

\(^{b}\) In element 109 at node 127, \( M_x = 811.88 \text{ in·lb/in} \) and \( M_y = 800.33 \text{ in·lb/in} \), the average of which is 806.1 in·lb/in.

\(^{c}\) In element 9 at node 10, \( M_y = 616.69 \text{ in·lb/in} \). This yields \( \sigma_{\text{bend}} = 616.69(6) = 3,700 \text{ psi} \).

**STAAD Input**

**Tip:** You can copy and paste this content directly into a .std file to run STAAD.Pro.
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Circular Surface Displacements and Stresses.STD is typically installed with the program.

STAAD SPACE: A CIRCULAR PLATE-FIXED ALONG ITS PERIMETER

START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* REFERENCE: TIMOSHENKO, S., STRENGTH OF MATERIALS
  PART II, ADVANCED THEORY AND PROBLEMS
  PAGES 96, 97, AND 103.

INPUT WIDTH 79
UNIT INCHES POUND

JOINT COORDINATES

ELEMENT INCIDENCES SHELL

7 7 8 26 25; 8 8 9 27 26; 9 9 10 28 27; 10 10 11 29 28; 11 11 12 30 29; 12 12 13 31 30; 13 13 14 32 31; 14 14 15 33 32; 15 15 16 34 33; 16 16 17 35 34; 17 17 18 36 35; 18 18 19 37 36; 19 19 20 38 37; 20 20 21 39 38; 21 21 22 40 39; 22 22 23 41 40; 23 23 24 42 41; 24 24 25 43 42; 25 25 26 44 43; 26 26 27 45 44;
ELEMENT PROPERTY
1 TO 126 THICKNESS 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 18 FIXED
LOAD 1 UNIFORM LOAD
ELEMENT LOAD
1 TO 126 PR GY -6
LOAD 2 POINT LOAD
JOINT LOAD
127 FY -7539.82
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 127
LOAD LIST 1
PRINT ELEMENT JOINT STRESSES LIST 9 109
LOAD LIST 2
PRINT ELEMENT JOINT STRESSES LIST 9
FINISH

**STAAD Output**

**JOINT DISPLACEMENT (INCH RADIANS)**  STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>127</td>
<td>0.00000</td>
<td>-0.08618</td>
<td>0.00000</td>
<td>-0.00000</td>
<td>0.00000</td>
<td>-0.00000</td>
</tr>
<tr>
<td>2</td>
<td>0.00000</td>
<td>-0.08646</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.00001</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT **************

93. LOAD LIST 1
94. PRINT ELEMENT JOINT STRESSES LIST 9 109

**ELEMENT JOINT STRESSES LIST**

:A CIRCULAR PLATE-FIXED ALONG ITS PERIMETER  -- PAGE NO.

**ELEMENT STRESSES**  FORCE,LENGTH UNITS= POUN INCH

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
<th>VONT</th>
<th>VONB</th>
<th>SX</th>
<th>SY</th>
<th>SXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>1</td>
<td>0.00</td>
<td>-103.62</td>
<td>115.36</td>
<td>719.90</td>
<td>0.01</td>
<td>4018.27</td>
<td>4018.27</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4319.39</td>
<td>4319.39</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>4319.39</td>
<td>692.18</td>
<td>1813.61</td>
<td>ANGLE= 90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>-692.18</td>
<td>-4319.39</td>
<td>1813.61</td>
<td>ANGLE= -0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td></td>
<td>14.39</td>
<td>-113.81</td>
<td>338.29</td>
<td>1147.77</td>
<td>123.92</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>6263.05</td>
<td>6263.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>6997.90</td>
<td>1918.50</td>
<td>2539.70</td>
<td>ANGLE= 81.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>-1918.50</td>
<td>-6997.90</td>
<td>2539.70</td>
<td>ANGLE= -8.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td></td>
<td>-14.38</td>
<td>-113.83</td>
<td>338.35</td>
<td>1147.91</td>
<td>-123.91</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>6263.75</td>
<td>6263.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>6998.72</td>
<td>1918.87</td>
<td>2539.93</td>
<td>ANGLE= -81.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>-1918.87</td>
<td>-6998.72</td>
<td>2539.93</td>
<td>ANGLE= 8.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td></td>
<td>-10.79</td>
<td>-93.43</td>
<td>-124.84</td>
<td>381.27</td>
<td>-175.70</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>3292.73</td>
<td>3292.73</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>2617.74</td>
<td>-1079.14</td>
<td>1848.44</td>
<td>ANGLE= -72.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>1079.14</td>
<td>-2617.74</td>
<td>1848.44</td>
<td>ANGLE= 17.4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td></td>
<td>10.79</td>
<td>-93.42</td>
<td>-124.84</td>
<td>381.32</td>
<td>175.75</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>3293.22</td>
<td>3293.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>2618.18</td>
<td>-1079.25</td>
<td>1848.71</td>
<td>ANGLE= 72.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>1079.25</td>
<td>-2618.18</td>
<td>1848.71</td>
<td>ANGLE= -17.4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>109</td>
<td>1</td>
<td>6.37</td>
<td>1.93</td>
<td>-799.52</td>
<td>-799.67</td>
<td>-0.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>-4796.24</td>
<td>-4799.94</td>
<td>1.35</td>
<td>ANGLE= -35.3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>4799.94</td>
<td>4796.24</td>
<td>1.35</td>
<td>ANGLE= 54.7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td></td>
<td>6.37</td>
<td>1.93</td>
<td>-811.88</td>
<td>-800.33</td>
<td>-0.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>127</td>
<td>4837.01</td>
<td>4837.01</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>-4871.30</td>
<td>34.67</td>
<td>ANGLE= -88.9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>4871.30</td>
<td>4801.97</td>
<td>34.67</td>
<td>ANGLE= 1.1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td></td>
<td>6.37</td>
<td>1.93</td>
<td>-792.77</td>
<td>-800.33</td>
<td>-0.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>109</td>
<td>4779.46</td>
<td>4779.46</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>-4802.02</td>
<td>22.72</td>
<td>ANGLE= -1.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>4802.02</td>
<td>4756.58</td>
<td>22.72</td>
<td>ANGLE= 88.4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Print element joint stresses list**

**Verification Examples**

V.04 Plate and Shell Elements

STAAD.Pro

3330

User Manual
### Verification Examples

#### V.04 Plate and Shell Elements

**STAAD.Pro 3331 User Manual**

---

#### Joint Stresses

<table>
<thead>
<tr>
<th>Joint</th>
<th>6.37</th>
<th>1.93</th>
<th>-793.92</th>
<th>-798.36</th>
<th>-0.21</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>-4763.47</td>
<td>-4790.19</td>
<td>13.36</td>
<td>ANGLE= 87.3</td>
<td></td>
</tr>
<tr>
<td>Bottom</td>
<td>4790.19</td>
<td>4763.47</td>
<td>13.36</td>
<td>ANGLE= -2.7</td>
<td></td>
</tr>
</tbody>
</table>

---

#### Maximum Stresses Among Selected Plates and Cases

<table>
<thead>
<tr>
<th>Plate No</th>
<th>9</th>
<th>9</th>
<th>9</th>
<th>109</th>
<th>109</th>
</tr>
</thead>
</table>

---

#### Element Stresses

<table>
<thead>
<tr>
<th>Element</th>
<th>Load</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>2</td>
<td>0.00</td>
<td>-34.32</td>
<td>75.98</td>
<td>484.41</td>
<td>0.01</td>
</tr>
</tbody>
</table>

---

#### Verification Examples

**STAAD.Pro 3331 User Manual**
V. Warped Surface Displacements

To find the displacements at the free end of a warped cantilever plate due to in-plane load and out of plane loads.

Reference


Problem

The finite element model is as shown below: Find the displacements at the tip in the direction of the loads. Loading is unit forces at the free end: in plane and out of plane.

- $E = 29,000.0$ ksi.
- $L = 12.0$ in.
- $B = 1.1$ in.
- $t = 0.22$ in.
- Twist = 90º (root to tip)
- Poisson’s ratio = 0.22

![Figure 393: Finite element model of warped, cantilever plate](image)

Comparison

**Table 408: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta$ due to in-plane load (in)</td>
<td>$5.424(10)^{-3}$</td>
<td>$5.590(10)^{-3}$</td>
<td>3.1%</td>
<td>Instead of using triangular element, a more advanced element type could to be used. Also the mesh size could be reduced to get closer result in comparison the theoretical value.</td>
</tr>
<tr>
<td>$\delta$ due to out-of-plane load (in)</td>
<td>$1.754(10)^{-3}$</td>
<td>$1.950(10)^{-3}$</td>
<td>11.2%</td>
<td></td>
</tr>
</tbody>
</table>
STAAD Input

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Warped Surface Displacements.STD is typically installed with the program.

**STAAD SPACE : A WARPED CANTILEVER PLATE**

START JOB INFORMATION
ENGINER DATE 14-Sep-18
END JOB INFORMATION

* * REFERENCE: MACNEAL AND HARDER, A PROPOSED STANDARD SET OF PROBLEMS
* TO TEST FINITE ELEMENT ACCURACY,
* FINITE ELEMENT IN ANALYSIS AND DESIGN 1, NORTH HOLLAND
* 1985

INPUT WIDTH 72
UNIT INCHES POUND

JOINT COORDINATES
1 0 -0.55 0; 2 1 -0.545 -0.072; 3 2 -0.531 -0.142; 4 3 -0.508 -0.21;
5 4 -0.476 -0.275; 6 5 -0.436 -0.335; 7 6 -0.389 -0.389;
8 7 -0.335 -0.436; 9 8 -0.275 -0.476; 10 9 -0.21 -0.508;
11 10 -0.142 -0.531; 12 11 -0.072 -0.545; 13 12 0 -0.55; 14 0 0 0;
15 1 0 0; 16 2 0 0; 17 3 0 0; 18 4 0 0; 19 5 0 0; 20 6 0 0; 21 7 0 0;
22 8 0 0; 23 9 0 0; 24 10 0 0; 25 11 0 0; 26 12 0 0; 27 0 0.55 0;
28 1 0.545 0.072; 29 2 0.531 0.142; 30 3 0.508 0.21; 31 4 0.476 0.275;
32 5 0.436 0.335; 33 6 0.389 0.389; 34 7 0.335 0.436; 35 8 0.275 0.476;
36 9 0.21 0.508; 37 10 0.142 0.531; 38 11 0.072 0.545; 39 12 0 0.55;

ELEMENT INCIDENCES SHELL
1 1 2 15; 2 15 14 1; 3 14 15 28; 4 28 27 14; 5 2 3 16; 6 16 15 2;
7 15 16 29; 8 29 28 15; 9 3 4 17; 10 17 16 3; 11 16 17 30; 12 30 29 16;
13 4 5 18; 14 18 17 4; 15 17 18 31; 16 31 30 17; 17 5 6 19; 18 19 18 5;
19 18 19 32; 20 32 31 18; 21 6 7 20; 22 20 19 6; 23 19 20 33;
24 33 32 19; 25 7 8 21; 26 21 20 7; 27 20 21 34; 28 34 33 20; 29 8 9 22;
30 22 21 8; 31 21 22 35; 32 35 34 21; 33 9 10 23; 34 23 22 9;
35 22 23 36; 36 36 35 22; 37 36 37 24; 38 24 23 36; 39 23 24 11;
40 11 10 23; 41 37 38 25; 42 25 24 37; 43 24 25 12; 44 12 11 24;
45 38 39 26; 46 26 25 38; 47 25 26 13; 48 13 12 25;

ELEMENT PROPERTY
1 TO 48 THICKNESS 0.32
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.9e+07
POISSON 0.22
END DEFINE MATERIAL

CONSTANTS
MATERIAL MATERIAL1 ALL

SUPPORTS
1 14 27 FIXED

LOAD 1 UNIT LOAD AT TIP, OUT OF PLANE
JOINT LOAD
13 39 FY 0.25
26 FY 0.5

LOAD 2 UNIT LOAD AT TIP, IN PLANE
JOINT LOAD
V. Curved Roof Displacements and Stresses

A Cylindrical roof is supported along two circular edges. Using plate/shell elements, find the vertical deflection at the center of the free edge, principal stresses at the center of the support and center of the free edge (top and bottom of the roof plate) due to uniformly distributed gravity load.

Reference


Problem

For the cylindrical roof shell calculate the following deflection and stresses due to the gravity load.

The vertical deflection, $\delta_y$, at the center of the free edge.

Principal stresses, $\sigma_{max}$ and $\sigma_{min}$, at the center line section at the vertical angle (top and bottom of the roof plate element). Principal stresses, $\sigma_{max}$ and $\sigma_{min}$, at the center section of the free edge (top and bottom of the roof plate element).

$$ E = 4.32 \times (10)^8 \text{ psf} $$

$$ t = 3.0 \text{ in.} $$

Poisson's ratio = 0.0 in theory (0.01001 in STAAD)

$$ w = 90 \text{ psf (uniform on surface).} $$

$$ L = 50 \text{ feet.} $$

$$ r = 25 \text{ feet, 40º sector either side of vertical} $$

Boundary conditions: simply supported on circular edges
Comparison

Table 409: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta_y$, at the center of the free edge (in) (y translation at node 63)</td>
<td>-3.703</td>
<td>3.650</td>
<td>1.4%</td>
<td>A finer mesh may reduce the difference between the theoretical and software output.</td>
</tr>
<tr>
<td>$\sigma$ at center (ft-kip)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bottom</td>
<td>191.23</td>
<td>181.55a</td>
<td>5.1%</td>
<td></td>
</tr>
<tr>
<td>Top</td>
<td>218.74</td>
<td>209.16a</td>
<td>4.4%</td>
<td></td>
</tr>
<tr>
<td>$\sigma$ at free edge (ft-kip)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bottom</td>
<td>215.57</td>
<td>208.33b</td>
<td>3.4%</td>
<td></td>
</tr>
<tr>
<td>Top</td>
<td>340.7</td>
<td>334.95b</td>
<td>1.7%</td>
<td></td>
</tr>
</tbody>
</table>

a. Plate 97, Joint 55, SMAX and SMIN
b. Plate 111, Joint 63, SMAX

STAAD Input

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.
Verification Examples
V.04 Plate and Shell Elements

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Curved Roof Displacements and Stresses.STD is typically installed with the program.

STAAD SPACE SCORDELIS-LO ROOF
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* A CYLINDRICAL ROOF SUPPORTED ALONG TWO CIRCULAR EDGES


* Theory displacement = .3086 feet or 3.7032 inches
* STAAD displacement with POIS = 0.30
  Joint 63&173 -3.79217 inches
  Joint 63&173  2.01523 -3.79217  0.00009  -0.00010  0.00019 0.03358
* STAAD displacement with POIS = 0.01001
  Joint 63&173 -3.65022 inches
  Joint 63&173  1.92962 -3.65022  0.00009  -0.00008  0.00015 0.03148

INPUT WIDTH 72
UNIT FEET POUND
JOINT COORDINATES
1 0 25 0; 2 2.179 24.905 0; 3 4.341 24.62 0; 4 6.471 24.148 0;
5 8.551 23.492 0; 6 10.566 22.651 0; 7 12.5 21.651 0; 8 14.339 20.479 0;
90 16.07 19.151 37.503; 91 0 25 41.67; 92 2.179 24.905 41.67; 93 4.341 24.62 41.67; 94 6.471 24.148 41.67;
95 8.551 23.492 41.67; 96 10.566 22.658 41.67; 97 12.5 21.651 41.67; 98 14.339 20.479 41.67;
### Verification Examples

#### V.04 Plate and Shell Elements

<table>
<thead>
<tr>
<th>Node</th>
<th>X Coordinate</th>
<th>Y Coordinate</th>
<th>Z Coordinate</th>
</tr>
</thead>
<tbody>
<tr>
<td>99</td>
<td>16.07</td>
<td>19.151</td>
<td>41.67</td>
</tr>
<tr>
<td>100</td>
<td>0</td>
<td>25</td>
<td>45.837</td>
</tr>
<tr>
<td>101</td>
<td>2.179</td>
<td>24.905</td>
<td>50.004</td>
</tr>
<tr>
<td>102</td>
<td>4.341</td>
<td>24.62</td>
<td>50.004</td>
</tr>
<tr>
<td>103</td>
<td>6.471</td>
<td>24.148</td>
<td>50.004</td>
</tr>
<tr>
<td>104</td>
<td>8.551</td>
<td>23.492</td>
<td>50.004</td>
</tr>
<tr>
<td>105</td>
<td>10.566</td>
<td>22.658</td>
<td>50.004</td>
</tr>
<tr>
<td>106</td>
<td>12.5</td>
<td>21.651</td>
<td>50.004</td>
</tr>
<tr>
<td>107</td>
<td>14.339</td>
<td>20.479</td>
<td>50.004</td>
</tr>
</tbody>
</table>

#### Additional Data

- STAAD.Pro 3337 User Manual
Verification Examples

V.04 Plate and Shell Elements
ELEMENT PROPERTY
1 TO 384 THICKNESS 0.25
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 432000
POISSON 0.01001
DENSITY 0.36
END DEFINE MATERIAL
UNIT FEET POUND
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 9 109 TO 125 214 TO 221 FIXED BUT FX MX MY
55 FIXED BUT FX FY MX MY MZ KFZ 12
LOAD 1 DEAD LOAD (SELFWEIGHT OF 90 PSF)
SELFWEIGHT Y -1
PERFORM ANALYSIS
UNIT FEET KIP
PRINT JOINT DISPLACEMENTS LIST 55 63 173
PRINT ELEMENT JOINT STRESSES LIST 97 111
FINISH
### JOINT DISPLACEMENT (INCH RADIANS)

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>55</td>
<td>1</td>
<td>0.0000</td>
<td>0.54458</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>63</td>
<td>1</td>
<td>-1.92962</td>
<td>-3.65022</td>
<td>0.00009</td>
<td>-0.00008</td>
<td>-0.00015</td>
<td>-0.03148</td>
</tr>
<tr>
<td>173</td>
<td>1</td>
<td>1.92962</td>
<td>-3.65022</td>
<td>0.00009</td>
<td>-0.00008</td>
<td>0.00015</td>
<td>0.03148</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

---

### ELEMENT STRESSES

**FOR FORCE,LENGTH UNITS= KIP FEET**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>97</td>
<td>1</td>
<td>0.14</td>
<td>0.06</td>
<td>-2.01</td>
<td>-0.10</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200.58</td>
<td>176.28</td>
<td>-14.22</td>
<td>-4.43</td>
<td>1.22</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>-13.63</td>
<td>SMIN=</td>
<td>-207.05</td>
<td>TMAX=</td>
<td>96.71</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>178.61</td>
<td>SMIN=</td>
<td>4.77</td>
<td>TMAX=</td>
<td>86.92</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.14</td>
<td>0.06</td>
<td>-2.03</td>
<td>-0.12</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>55</td>
<td></td>
<td>202.01</td>
<td>178.11</td>
<td>-13.81</td>
<td>-4.02</td>
<td>1.22</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>-15.15</td>
<td>SMIN=</td>
<td>-209.16</td>
<td>TMAX=</td>
<td>97.00</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>181.55</td>
<td>SMIN=</td>
<td>7.10</td>
<td>TMAX=</td>
<td>87.22</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.14</td>
<td>0.06</td>
<td>-1.96</td>
<td>-0.12</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>56</td>
<td></td>
<td>193.92</td>
<td>171.12</td>
<td>-13.81</td>
<td>-5.25</td>
<td>1.22</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>-16.38</td>
<td>SMIN=</td>
<td>-201.59</td>
<td>TMAX=</td>
<td>92.61</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>173.98</td>
<td>SMIN=</td>
<td>5.88</td>
<td>TMAX=</td>
<td>84.05</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.14</td>
<td>0.06</td>
<td>-2.03</td>
<td>-0.06</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>64</td>
<td></td>
<td>205.87</td>
<td>179.66</td>
<td>-15.04</td>
<td>-4.02</td>
<td>1.22</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>-9.36</td>
<td>SMIN=</td>
<td>-210.40</td>
<td>TMAX=</td>
<td>100.52</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>334.95</td>
<td>SMIN=</td>
<td>1.31</td>
<td>TMAX=</td>
<td>89.50</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.27</td>
<td>-0.24</td>
<td>0.05</td>
<td>0.67</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>111</td>
<td></td>
<td>326.36</td>
<td>218.90</td>
<td>-15.62</td>
<td>254.40</td>
<td>-16.47</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>310.89</td>
<td>SMIN=</td>
<td>-15.47</td>
<td>TMAX=</td>
<td>163.18</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>200.52</td>
<td>SMIN=</td>
<td>-18.38</td>
<td>TMAX=</td>
<td>109.45</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.27</td>
<td>-0.24</td>
<td>-0.09</td>
<td>0.67</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>62</td>
<td></td>
<td>296.45</td>
<td>162.76</td>
<td>-8.74</td>
<td>222.47</td>
<td>-16.47</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>287.08</td>
<td>SMIN=</td>
<td>-17.93</td>
<td>TMAX=</td>
<td>152.51</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>161.26</td>
<td>SMIN=</td>
<td>-2.96</td>
<td>TMAX=</td>
<td>82.11</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.27</td>
<td>-0.24</td>
<td>0.05</td>
<td>0.67</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>63</td>
<td></td>
<td>336.92</td>
<td>216.84</td>
<td>-8.74</td>
<td>270.37</td>
<td>-16.47</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>334.95</td>
<td>SMIN=</td>
<td>-3.91</td>
<td>TMAX=</td>
<td>169.43</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>208.33</td>
<td>SMIN=</td>
<td>-16.10</td>
<td>TMAX=</td>
<td>112.22</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.27</td>
<td>-0.24</td>
<td>0.05</td>
<td>0.42</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>72</td>
<td></td>
<td>323.64</td>
<td>252.42</td>
<td>-29.38</td>
<td>270.37</td>
<td>-16.47</td>
</tr>
<tr>
<td>TOP :</td>
<td>SMAX=</td>
<td>310.66</td>
<td>SMIN=</td>
<td>-24.56</td>
<td>TMAX=</td>
<td>167.61</td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX=</td>
<td>232.26</td>
<td>SMIN=</td>
<td>-36.38</td>
<td>TMAX=</td>
<td>134.32</td>
</tr>
</tbody>
</table>

SCORDELIS-LO ROOF -- PAGE NO.

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

---

**Verification Examples**

V.04 Plate and Shell Elements

STAAD.Pro 3340 User Manual
V. Spherical Shell Displacements

To find the displacement in the direction of the load due to a unit load applied at the quadrants of a quarter of a spherical shell.

Reference

Problem
For the quarter of a spherical shell find the displacement in the direction of the load.

\[ E = 6.825 \times 10^7 \text{ psi} \]
\[ \text{Poisson's ratio} = 0.3 \]
\[ t = 0.04 \text{ inches} \]
\[ r = 10 \text{ in.} \]

Unit forces on quadrants

Boundary conditions:
- Vertical restraint at center of free edge
- Symmetry defines boundary conditions

<table>
<thead>
<tr>
<th></th>
<th>MAXIMUM PRINCIPAL STRESS</th>
<th>MINIMUM PRINCIPAL STRESS</th>
<th>MAXIMUM SHEAR STRESS</th>
<th>MAXIMUM VONMISES STRESS</th>
<th>MAXIMUM TRESCA STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLATE NO.</td>
<td>111</td>
<td>111</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>CASE NO.</td>
<td>1</td>
<td>97</td>
<td>111</td>
<td>111</td>
<td>111</td>
</tr>
</tbody>
</table>
Comparison

Table 410: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection, δ (in)</td>
<td>0.094</td>
<td>0.09342</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

**STAAD Input**

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Spherical Shell Displacements.STD is typically installed with the program.

STAAD SPACE : A QUARTER OF A SPHERICAL SHELL
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* * REFERENCE: MACNEAL AND HARDER, A PROPOSED STANDARD SET OF PROBLEMS
* TO TEST FINITE ELEMENT ACCURACY,
* FINITE ELEMENT IN ANALYSIS AND DESIGN 1, NORTH HOLLAND
* 1985
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 10 0 0; 2 0 0 10; 3 1.951 0 9.808; 4 3.827 0 9.239; 5 5.556 0 8.315;
6 7.071 0 7.071; 7 8.315 0 5.556; 8 9.239 0 3.827; 9 9.808 0 1.951;
10 9.877 1.564 0; 11 0 1.564 9.877; 12 1.927 1.564 9.687;
19 9.511 3.09 0; 20 0 3.09 9.511; 21 1.856 3.09 9.328;
22 3.64 3.09 8.787; 23 5.284 3.09 7.908; 24 6.725 3.09 6.725;
25 7.908 3.09 5.284; 26 8.787 3.09 3.64; 27 9.328 3.09 1.855;
28 8.91 4.54 0; 29 0 4.54 8.91; 30 1.738 4.54 8.739;
31 3.41 4.54 8.232; 32 4.95 4.54 7.408; 33 6.3 4.54 6.3;
34 7.408 4.54 4.95; ...
V. 2D Circular Plate In-Plane Stresses

A thick cylindrical plate supported along 2 radial edges. Find the radial displacement, radial stress, tangential stress and longitudinal stress at inner surface due to a unit pressure applied at the inner surface.

Reference


Problem

Loading is unit pressure at inner radius

\[ E = 1 \times (10)^6 \text{ psi} \]

Poisson’s ratio = 0.3

Inner radius = 3.0 in

Outer radius = 9.0 in
Figure 396: Semi-circular plate finite element model

Comparison

Table 411: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radial deflection (10^{-3} in)</td>
<td>4.582</td>
<td>4.650^a</td>
<td>1.5%</td>
</tr>
<tr>
<td>Radial stress (psi)</td>
<td>-1.00</td>
<td>-0.95^b</td>
<td>5.0%</td>
</tr>
<tr>
<td>Tangential stress (psi)</td>
<td>1.25</td>
<td>1.26^b</td>
<td>0.8%</td>
</tr>
</tbody>
</table>
### Result Type

<table>
<thead>
<tr>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.075</td>
<td>0.080</td>
<td>7.0%</td>
</tr>
</tbody>
</table>

a. Radial displacements are measured along FY at node 102 and FX at node 101.

b. On element 82, at node 102, SX is tangential stress, SY is radial stress.

---

**STAAD Input**

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Circular Plate In-Plane Stresses.STD is typically installed with the program.

STAAD SPACE: A THICK WALLED CYLINDER PLATE SUPPORTED ALONG 2 RADIAL EDGES

**START JOB INFORMATION**

ENGINEER DATE 14-Sep-18

**END JOB INFORMATION**

* 
* REFERENCE: MACNEAL AND HARDER, A PROPOSED STANDARD SET OF PROBLEMS
* TO TEST FINITE ELEMENT ACCURACY,
* 
* FINITE ELEMENT IN ANALYSIS AND DESIGN 1, NORTH HOLLAND
* 
* 1985

**INPUT WIDTH 72**

UNIT INCHES POUND

JOINT COORDINATES

1 9 0 0; 2 0 9 0; 3 1.563 8.863 0; 4 3.078 8.457 0; 5 4.5 7.794 0;
6 5.785 6.894 0; 7 6.894 5.785 0; 8 7.794 4.5 0; 9 8.457 3.078 0;
10 8.863 1.563 0; 11 7.875 0 0; 12 0 7.875 0; 13 1.367 7.755 0;
14 2.693 7.4 0; 15 3.938 6.82 0; 16 5.062 6.033 0; 17 6.033 5.062 0;
18 6.82 3.937 0; 19 7.4 2.693 0; 20 7.755 1.367 0; 21 6.75 0 0;
22 0 6.75 0; 23 1.172 6.647 0; 24 2.309 6.343 0; 25 3.375 5.846 0;
26 4.339 5.171 0; 27 5.171 4.339 0; 28 5.846 3.375 0; 29 6.343 2.309 0;
30 6.647 1.172 0; 31 5.975 0 0; 32 0 5.975 0; 33 1.038 5.884 0;
34 2.044 5.615 0; 35 2.988 5.174 0; 36 3.841 4.577 0; 37 4.577 3.841 0;
38 5.175 2.987 0; 39 5.615 2.044 0; 40 5.884 1.038 0; 41 5.2 0 0;
42 0 5.2 0; 43 0.903 5.121 0; 44 1.779 4.886 0; 45 2.6 4.503 0;
46 3.343 3.983 0; 47 3.983 3.342 0; 48 4.503 2.6 0; 49 4.886 1.778 0;
50 5.121 0.903 0; 51 4.7 0 0; 52 0 4.7 0; 53 0.816 4.629 0;
54 1.608 4.417 0; 55 2.35 4.07 0; 56 3.021 3.6 0; 57 3.6 3.021 0;
58 4.07 2.35 0; 59 4.417 1.607 0; 60 4.629 0.816 0; 61 4.2 0 0;
62 0 4.2 0; 63 0.729 4.136 0; 64 1.436 3.947 0; 65 2.1 3.637 0;
66 2.7 3.217 0; 67 3.217 2.7 0; 68 3.637 2.1 0; 69 3.947 1.436 0;
70 4.136 0.729 0; 71 3.85 0 0; 72 0 3.85 0; 73 0.669 3.792 0;
74 1.317 3.618 0; 75 1.925 3.34 0; 76 2.475 2.949 0; 77 2.949 2.475 0;
78 3.344 1.925 0; 79 3.618 1.317 0; 80 3.792 0.669 0; 81 3.5 0 0;
82 0 3.5 0; 83 0.668 3.447 0; 84 1.197 3.289 0; 85 1.75 3.031 0;
86 2.25 2.681 0; 87 2.681 2.25 0; 88 3.031 1.75 0; 89 3.289 1.197 0;
90 3.447 0.668 0; 91 3.25 0 0; 92 0 3.25 0; 93 0.564 3.281 0;
94 1.112 3.054 0; 95 1.625 2.815 0; 96 2.089 2.49 0; 97 2.49 2.089 0;
98 2.815 1.625 0; 99 3.054 1.112 0; 100 3.281 0.564 0; 101 0 0 0;
102 0 0 0; 103 0.521 2.954 0; 104 1.026 2.819 0; 105 1.5 2.598 0;
106 1.928 2.298 0; 107 2.298 1.928 0; 108 2.598 1.5 0;
```plaintext
Verification Examples
V.04 Plate and Shell Elements

109 2.819 1.026 0; 110 2.954 0.521 0;
ELEMENT INCIDENCES SHELL
1 2 3 13 12; 2 3 4 14 13; 3 4 5 15 14; 4 5 6 16 15; 5 6 7 17 16;
6 7 8 18 17; 7 8 9 19 18; 8 9 10 20 19; 9 10 1 11 20; 10 12 13 23 22;
11 13 14 24 23; 12 14 15 25 24; 13 15 16 26 25; 14 16 17 27 26;
15 17 18 28 27; 16 18 19 29 28; 17 19 20 30 29; 18 20 11 21 30;
19 22 23 32 31; 20 23 24 33 32; 21 24 25 34 33; 22 25 26 35 34;
23 26 27 36 35; 24 27 28 37 36; 25 28 29 38 37; 26 29 30 40 39;
27 30 31 40 39; 28 31 32 42 41; 29 32 33 44 43; 30 33 34 45 44;
31 34 35 46 45; 32 35 36 47 46; 33 36 37 48 47; 34 37 38 49 48;
35 38 39 50 49; 36 39 40 51 50; 37 40 41 52 51; 38 41 42 53 52;
39 42 43 54 53; 40 43 44 55 54; 41 44 45 56 55; 42 45 46 57 56;
43 46 47 58 57; 44 47 48 59 58; 45 48 49 60 59; 46 49 50 61 60;
47 50 51 62 61; 48 51 52 63 62; 49 52 53 64 63; 50 53 54 65 64;
51 54 55 66 65; 52 55 56 67 66; 53 56 57 68 67; 54 57 58 69 68;
55 58 59 70 69; 56 59 60 71 70; 57 60 61 72 71; 58 61 62 73 72;
59 62 63 74 73; 60 63 64 75 74; 61 64 65 76 75; 62 65 66 77 76;
63 66 67 78 77; 64 67 68 79 78; 65 68 69 80 79; 66 69 70 81 80;
67 70 71 82 81; 68 71 72 83 82; 69 72 73 84 83; 70 73 74 85 84;
71 74 75 86 85; 72 75 76 87 86; 73 76 77 88 87; 74 77 78 89 88;
75 78 79 90 89; 76 79 80 91 90; 77 80 81 92 91; 78 81 82 93 92;
79 82 83 94 93; 80 83 84 95 94; 81 84 85 96 95; 82 85 86 97 96;
83 86 87 98 97; 84 87 88 99 98; 85 88 89 100 99; 86 89 90 101 100;
87 90 91 102 101; 88 91 92 103 102; 89 92 93 104 103; 90 93 94 105 104;
91 94 95 106 105; 92 95 96 107 106; 93 96 97 108 107; 94 97 98 109 108;
95 98 99 110 109; 96 99 100 110 110;
ELEMENT PROPERTY
1 TO 90 THICKNESS 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
2 12 22 42 52 62 72 82 92 102 FIXED BUT FY
1 11 21 31 41 51 61 71 81 91 101 FIXED BUT FX
LOAD 1 UNIT PRESSURE AT INNER RADIUS
JOINT LOAD
101 FX 0.2618
110 FX 0.5156 FY 0.0999
109 FX 0.492 FY 0.1791
108 FX 0.4534 FY 0.2618
107 FX 0.4011 FY 0.3366
106 FX 0.3366 FY 0.4011
105 FX 0.2618 FY 0.4534
104 FX 0.1791 FY 0.492
103 FX 0.0999 FY 0.5156
102 FY 0.2618
* CREATED LOAD 2 IN ORDER TO PRINT DISPLACEMENT VALUES
LOAD 2 MULTIPLY LOAD BY 1000
REPEAT LOAD
1 1000.0
PERFORM ANALYSIS
LOAD LIST 2
PRINT JOINT DISPLACEMENTS LIST 101 102 107
LOAD LIST 1
```
### STAAD Output

**Print Element Joint Stresses List 82 83 89 90**

**FINISH**

**Verification Examples**

V.04 Plate and Shell Elements

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>2</td>
<td>0.00465</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>102</td>
<td>0.00000</td>
<td>0.00465</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>107</td>
<td>2</td>
<td>0.00357</td>
<td>0.00299</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

97. Load List 1
98. Print Element Joint Stresses List 82 83 89 90

**Element Joint Stresses**

:A Thick Walled Cylinder Plate Supported Along 2 Radial -- Page No.

5

**Element Stresses**

FORCE, LENGTH UNITS = POUN INCH

<p>| STRESS = FORCE/UNIT WIDTH/THICK,  MOMENT = FORCE-LENGTH/UNIT WIDTH |
|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|</p>
<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>SX</th>
<th>SY</th>
<th>SX-Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>82</td>
<td>1</td>
<td>1.81</td>
<td>1.81</td>
<td>1.17</td>
<td>-0.91</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.08</td>
<td>2.08</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 1.17</td>
<td>SMIN= -0.91</td>
<td>TMAX= 1.04</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1.17</td>
<td>SMIN= -0.91</td>
<td>TMAX= 1.04</td>
<td>ANGLE= 0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT 92</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>1.69</td>
<td>1.69</td>
<td>1.07</td>
<td>-0.88</td>
<td>-0.08</td>
<td>0.00</td>
</tr>
<tr>
<td>TOP: SMAX= 1.07</td>
<td>SMIN= -0.88</td>
<td>TMAX= 0.98</td>
<td>ANGLE= -2.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1.07</td>
<td>SMIN= -0.88</td>
<td>TMAX= 0.98</td>
<td>ANGLE= -2.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT 93</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>1.69</td>
<td>1.69</td>
<td>1.07</td>
<td>-0.88</td>
<td>-0.08</td>
<td>0.00</td>
</tr>
<tr>
<td>TOP: SMAX= 1.07</td>
<td>SMIN= -0.88</td>
<td>TMAX= 0.98</td>
<td>ANGLE= -2.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1.07</td>
<td>SMIN= -0.88</td>
<td>TMAX= 0.98</td>
<td>ANGLE= -2.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT 103</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>1.93</td>
<td>1.93</td>
<td>1.26</td>
<td>-0.95</td>
<td>0.08</td>
<td>0.00</td>
</tr>
<tr>
<td>TOP: SMAX= 1.26</td>
<td>SMIN= -0.95</td>
<td>TMAX= 1.11</td>
<td>ANGLE= 2.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1.26</td>
<td>SMIN= -0.95</td>
<td>TMAX= 1.11</td>
<td>ANGLE= 2.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT 102</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>1.93</td>
<td>1.93</td>
<td>1.26</td>
<td>-0.95</td>
<td>0.08</td>
<td>0.00</td>
</tr>
<tr>
<td>TOP: SMAX= 1.26</td>
<td>SMIN= -0.95</td>
<td>TMAX= 1.11</td>
<td>ANGLE= 2.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1.26</td>
<td>SMIN= -0.95</td>
<td>TMAX= 1.11</td>
<td>ANGLE= 2.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>83</td>
<td>1</td>
<td>1.81</td>
<td>1.81</td>
<td>1.17</td>
<td>-0.92</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2.08</td>
<td>2.08</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 1.17</td>
<td>SMIN= -0.92</td>
<td>TMAX= 1.04</td>
<td>ANGLE= -0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1.17</td>
<td>SMIN= -0.92</td>
<td>TMAX= 1.04</td>
<td>ANGLE= -0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT 93</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>1.69</td>
<td>1.69</td>
<td>1.07</td>
<td>-0.88</td>
<td>-0.08</td>
<td>0.00</td>
</tr>
<tr>
<td>TOP: SMAX= 1.07</td>
<td>SMIN= -0.88</td>
<td>TMAX= 0.98</td>
<td>ANGLE= -2.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 1.07</td>
<td>SMIN= -0.88</td>
<td>TMAX= 0.98</td>
<td>ANGLE= -2.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT 94</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>1.69</td>
<td>1.69</td>
<td>1.06</td>
<td>-0.88</td>
<td>0.08</td>
<td>0.00</td>
</tr>
<tr>
<td>TOP : SMAX= 1.07</td>
<td>SMIN= -0.88</td>
<td>TMAX= 0.98</td>
<td>ANGLE= 2.3</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Verification Examples**

V.04 Plate and Shell Elements

---

<table>
<thead>
<tr>
<th>ELEMENT STRESSES</th>
<th>FORCE,LENGTH UNITS= POUN INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT</td>
<td>VONB</td>
<td>TRESSAT</td>
<td>TREScab</td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>99</td>
<td>1.69</td>
<td>1.69</td>
<td>1.06</td>
<td>-0.88</td>
<td>-0.08</td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>1.07</td>
<td>SMIN=</td>
<td>-0.88</td>
<td>TMAX=</td>
<td>0.98</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>100</td>
<td>1.69</td>
<td>1.69</td>
<td>1.07</td>
<td>-0.88</td>
<td>0.88</td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>1.07</td>
<td>SMIN=</td>
<td>-0.88</td>
<td>TMAX=</td>
<td>0.98</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>110</td>
<td>1.93</td>
<td>1.93</td>
<td>1.26</td>
<td>-0.95</td>
<td>0.08</td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>1.26</td>
<td>SMIN=</td>
<td>-0.96</td>
<td>TMAX=</td>
<td>1.11</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>109</td>
<td>1.93</td>
<td>1.93</td>
<td>1.26</td>
<td>-0.95</td>
<td>-0.08</td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>1.26</td>
<td>SMIN=</td>
<td>-0.96</td>
<td>TMAX=</td>
<td>1.11</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>90</td>
<td>1.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1.81</td>
<td>1.81</td>
<td>1.17</td>
<td>-0.92</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>1.07</td>
<td>SMIN=</td>
<td>-0.88</td>
<td>TMAX=</td>
<td>0.98</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>101</td>
<td>1.93</td>
<td>1.93</td>
<td>1.26</td>
<td>-0.95</td>
<td>0.08</td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>1.26</td>
<td>SMIN=</td>
<td>-0.95</td>
<td>TMAX=</td>
<td>1.11</td>
</tr>
<tr>
<td>JOINT</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>110</td>
<td>1.93</td>
<td>1.93</td>
<td>1.26</td>
<td>-0.95</td>
<td>-0.08</td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>1.26</td>
<td>SMIN=</td>
<td>-0.95</td>
<td>TMAX=</td>
<td>1.11</td>
</tr>
</tbody>
</table>

:A THICK WALLED CYLINDER PLATE SUPPORTED ALONG 2 RADIAL -- PAGE NO. 6

---

**STAAD.Pro** 3349 User Manual
V. 2D Rectangular Plate with fixed edges

To find the vertical deflection and bending moments at several points due to a unit pressure on a thin rectangular plate simply supported along 4 edges.

Reference

Problem
Loading is unit pressure (1 psi) over entire surface.

\[ E = 1 \times (10)^6 \text{ psi} \]

Poisson’s ratio = 0.3

Length = 16 in.

Width = 10 in.

Thickness = 0.2 in
### Comparison

#### Table 412: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical deflection, ( \delta ) (in) at joint</td>
<td>162</td>
<td>0.036</td>
<td>0.03549</td>
</tr>
<tr>
<td></td>
<td>166</td>
<td>0.113</td>
<td>0.11230</td>
</tr>
<tr>
<td></td>
<td>306</td>
<td>0.025</td>
<td>0.02478</td>
</tr>
<tr>
<td>Bending moment, ( M_x ) (in·lb) at joint</td>
<td>9</td>
<td>1.763</td>
<td>1.64</td>
</tr>
<tr>
<td></td>
<td>89</td>
<td>8.513</td>
<td>8.41</td>
</tr>
<tr>
<td></td>
<td>96</td>
<td>1.098</td>
<td>1.05</td>
</tr>
<tr>
<td>Bending moment, ( M_y ) (in·lb) at joint</td>
<td>9</td>
<td>0.897</td>
<td>0.86</td>
</tr>
<tr>
<td></td>
<td>89</td>
<td>4.873</td>
<td>4.81</td>
</tr>
<tr>
<td></td>
<td>96</td>
<td>1.108</td>
<td>0.99</td>
</tr>
</tbody>
</table>
**STAAD Input**

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Rectangular Plate with fixed edges.std is typically installed with the program.

**STAAD SPACE : UNIFORM PRESSURE ON RECTANGULAR PLATE ELEMENTS**

**START JOB INFORMATION**

**ENGINEER DATE 14-Sep-18**

**END JOB INFORMATION**

* FILE: 2D Rectangular Plate with fixed edges.STD


**UNIT INCHES POUND**

**JOINT COORDINATES**

1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0;
9 8 0 0; 10 9 0 0; 11 10 0 0; 21 0 0 1; 22 1 0 1; 23 2 0 1; 24 3 0 1;
25 4 0 1; 26 5 0 1; 27 6 0 1; 28 7 0 1; 29 8 0 1; 30 9 0 1; 31 10 0 1;
41 0 0 2; 42 1 0 2; 43 2 0 2; 44 3 0 2; 45 4 0 2; 46 5 0 2; 47 6 0 2;
48 7 0 2; 49 8 0 2; 50 9 0 2; 51 10 0 2; 61 0 0 3; 62 1 0 3; 63 2 0 3;
64 3 0 3; 65 4 0 3; 66 5 0 3; 67 6 0 3; 68 7 0 3; 69 8 0 3; 70 9 0 3;
71 10 0 3; 81 0 0 4; 82 1 0 4; 83 2 0 4; 84 3 0 4; 85 4 0 4; 86 5 0 4;
87 6 0 4; 88 7 0 4; 89 8 0 4; 90 9 0 4; 91 10 0 4; 101 0 0 5; 102 1 0 5;
103 2 0 5; 104 3 0 5; 105 4 0 5; 106 5 0 5; 107 6 0 5; 108 7 0 5;
109 8 0 5; 110 9 0 5; 111 10 0 5; 121 0 0 6; 122 1 0 6; 123 2 0 6;
124 3 0 6; 125 4 0 6; 126 5 0 6; 127 6 0 6; 128 7 0 6; 129 8 0 6;
130 9 0 6; 131 10 0 6; 141 0 0 7; 142 1 0 7; 143 2 0 7; 144 3 0 7;
145 4 0 7; 146 5 0 7; 147 6 0 7; 148 7 0 7; 149 8 0 7; 150 9 0 7;
151 10 0 7; 161 0 0 8; 162 1 0 8; 163 2 0 8; 164 3 0 8; 165 4 0 8;
166 5 0 8; 167 6 0 8; 168 7 0 8; 169 8 0 8; 170 9 0 8; 171 10 0 8;
181 0 0 9; 182 1 0 9; 183 2 0 9; 184 3 0 9; 185 4 0 9; 186 5 0 9;
187 6 0 9; 188 7 0 9; 189 8 0 9; 190 9 0 9; 191 10 0 9; 201 0 0 10;
202 1 0 10; 203 2 0 10; 204 3 0 10; 205 4 0 10; 206 5 0 10; 207 6 0 10;
208 7 0 10; 209 8 0 10; 210 9 0 10; 211 10 0 10; 221 0 0 11; 222 1 0 11;
223 2 0 11; 224 3 0 11; 225 4 0 11; 226 5 0 11; 227 6 0 11; 228 7 0 11;
229 8 0 11; 230 9 0 11; 231 10 0 11; 241 0 0 12; 242 1 0 12; 243 2 0 12;
244 3 0 12; 245 4 0 12; 246 5 0 12; 247 6 0 12; 248 7 0 12; 249 8 0 12;
250 9 0 12; 251 10 0 12; 261 0 0 13; 262 1 0 13; 263 2 0 13; 264 3 0 13;
265 4 0 13; 266 5 0 13; 267 6 0 13; 268 7 0 13; 269 8 0 13; 270 9 0 13;
271 10 0 13; 281 0 0 14; 282 1 0 14; 283 2 0 14; 284 3 0 14; 285 4 0 14;
286 5 0 14; 287 6 0 14; 288 7 0 14; 289 8 0 14; 290 9 0 14; 291 10 0 14;
301 0 0 15; 302 1 0 15; 303 2 0 15; 304 3 0 15; 305 4 0 15; 306 5 0 15;
307 6 0 15; 308 7 0 15; 309 8 0 15; 310 9 0 15; 311 10 0 15; 312 0 0 16;
321 1 0 16; 322 2 0 16; 323 3 0 16; 324 4 0 16; 325 5 0 16; 326 6 0 16;
328 7 0 16; 329 8 0 16; 330 9 0 16; 331 10 0 16;

**ELEMENT INCIDENCES SHELL**

1 1 2 22 21; 2 21 22 42 41; 3 41 42 62 61; 4 61 62 82 81;
5 81 82 102 101; 6 101 102 122 121; 7 121 122 142 141;
8 141 142 162 161; 9 161 162 182 181; 10 181 182 202 201;
11 201 202 222 221; 12 221 222 242 241; 13 241 242 262 261;
14 261 262 282 281; 15 281 282 302 301; 16 301 302 322 321;
ELEMENT PROPERTY
1 TO 16 21 TO 36 41 TO 56 61 TO 76 81 TO 96 THICKNESS 0.2
101 TO 116 121 TO 136 141 TO 156 161 TO 176 181 TO 196 THICKNESS 0.2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 11 321 331 FIXED
2 TO 10 322 TO 330 FIXED BUT MX

Verification Examples
V.04 Plate and Shell Elements
LOAD 1 UNIFORM PRESSURE

PRINT JOINT DISPLACEMENTS LIST 162 166 306
PRINT ELEMENT STRESSES LIST 9 89 96
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTA N</th>
<th>Y-ROTA N</th>
<th>Z-ROTA N</th>
</tr>
</thead>
<tbody>
<tr>
<td>162</td>
<td>0.00000</td>
<td>-0.03549</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.03420</td>
</tr>
<tr>
<td>166</td>
<td>0.00000</td>
<td>-0.11230</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>306</td>
<td>0.00000</td>
<td>-0.02478</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

PRINT ELEMENT STRESSES LIST 9 89 96

<table>
<thead>
<tr>
<th>ELEMENT STRESSES</th>
<th>FORCE,LENGTH UNITS= POUN INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH</td>
<td></td>
</tr>
<tr>
<td>SQX</td>
<td>SQY</td>
</tr>
<tr>
<td>TRECSCAT</td>
<td>TRECSCAB</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
</tr>
<tr>
<td>225.14</td>
<td>225.14</td>
</tr>
<tr>
<td>259.42</td>
<td>259.42</td>
</tr>
</tbody>
</table>

TOP : SMAX= 259.42 SMIN= -115.15 TMAX= 72.14 ANGLE= 17.6
BOTT: SMAX= -115.15 SMIN= -259.42 TMAX= 72.14 ANGLE= -72.4

89   | 1   | 1.99 | -0.51 | 8.41 | 4.81 | 0.04 |
| 1096.84 | 1096.84 | 0.00 | 0.00 | 0.00 |
| 1262.19 | 1262.19 | 0.00 | 0.00 | 0.00 |

TOP : SMAX= 1262.19 SMIN= 721.80 TMAX= 270.20 ANGLE= 0.7
BOTT: SMAX= -721.80 SMIN= -1262.19 TMAX= 270.20 ANGLE= -89.3

96   | 1   | 0.25 | -15.87 | 1.05 | 0.99 | 0.58 |
| 214.93 | 214.93 | 0.00 | 0.00 | 0.00 |
| 240.24 | 240.24 | 0.00 | 0.00 | 0.00 |

TOP : SMAX= 240.24 SMIN= 66.20 TMAX= 87.02 ANGLE= 43.5
BOTT: SMAX= -66.20 SMIN= -240.24 TMAX= 87.02 ANGLE= -46.5

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

<table>
<thead>
<tr>
<th>PLATE NO.</th>
<th>89</th>
<th>89</th>
<th>89</th>
<th>89</th>
<th>89</th>
</tr>
</thead>
<tbody>
<tr>
<td>CASE NO.</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
V. 2D Plate Thermal Moment and Stress

To find the bending moment due to thermal load through the thickness of a square plate.

Reference


Problem

Temperature varies 100°F linearly through the thickness of a square plate that is fixed on the edges. Calculate the bending moment on the edges and the maximum bending stress.

\[ E = 30,000.0 \text{ ksi} \]
\[ \alpha = 70 \times (10)^{-7} \text{ in/in/°F} \]

Size = 5” × 5”

Thickness = 0.5 in

Poisson’s ratio = 0.3

*Figure 398: Model*
Comparison

Table 413: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment (in·lb/in)</td>
<td>625.0</td>
<td>625.0</td>
<td>none</td>
</tr>
<tr>
<td>Maximum stress (psi)</td>
<td>15,000</td>
<td>15,000</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

Tip: You can copy and paste this content directly into a .std file to run STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Plate Thermal Moment and Stress.STD is typically installed with the program.

```
STAAD SPACE : THERMAL LOADING OF A PLATE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

REFERENCE: TIMOSHENKO, S., "STRENGTH OF MATERIALS," 3RD EDITION,
             D. VAN NOSTRAND CO., 1956.

UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 5 0 0; 3 0 5 0; 4 5 5 0;
ELEMENT INCIDENCES SHELL
1 1 2 4 3;
ELEMENT PROPERTY
1 THICKNESS 0.5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
ALPHA 7e-06
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 4 FIXED
LOAD 1 NON UNIFORM HEATING OF THE PLATE
TEMPERATURE LOAD
1 TEMP 0 100
PERFORM ANALYSIS
PRINT ELEMENT JOINT STRESSES ALL
FINISH
```
V. 2D Surface Displacements

To find the vertical deflection and bending moments due to an unit pressure in a rectangular plate simply supported along four edges.

References


Problem

Find the vertical deflection at the points shown in the sketch and the bending stress at the center of a 20 in. square plate subjected to a uniform pressure of 1 ksi. Use a quarter of the plate assuming proper boundary conditions along the lines of symmetry.

\[
E = 10,000.0 \text{ ksi} \\
\text{Thickness} = 5 \text{ in} \\
P\text{oisson's ratio} = 0.4
\]

Comparison

Table 414: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical deflections due to unit pressure, (10^{-3}) in</td>
<td>(\delta_1) 6.826</td>
<td>6.864</td>
<td>0.6%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>(\delta_9) 2.322</td>
<td>2.343</td>
<td>0.9%</td>
<td></td>
</tr>
</tbody>
</table>
### Result Type

<table>
<thead>
<tr>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>δ&lt;sub&gt;73&lt;/sub&gt;</td>
<td>2.684</td>
<td>2.707</td>
<td>0.9%</td>
</tr>
<tr>
<td>Maximum bending stress due to unit pressure, σ, element 1 (psi)</td>
<td>5,071</td>
<td>4,908</td>
<td>3.2%</td>
</tr>
</tbody>
</table>

The theoretical results from classical Plate theory was compared with the results from Finite Element model - hence the difference in results.

---

### STAAD Input

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Surface Displacements.STD is typically installed with the program.

STAAD SPACE : A SIMPLY SUPPORTED PLATE WITH TRIANGULAR ELEMENTS

START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0; 9 8 0 0; 10 9 0 0; 11 1 0 0; 12 2 0 0; 13 1 0 0; 14 2 0 0; 15 3 0 0; 16 4 0 1; 17 5 0 1; 18 6 0 1; 19 7 0 1; 20 8 0 1; 21 9 0 1; 22 10 0 1; 23 0 0 2; 24 1 0 2; 25 2 0 2; 26 3 0 2; 27 4 0 2; 28 5 0 2; 29 6 0 2; 30 7 0 2; 31 8 0 2; 32 9 0 2; 33 10 0 2; 34 1 0 3; 35 2 0 3; 36 3 0 3; 37 4 0 3; 38 5 0 3; 39 6 0 3; 40 7 0 3; 41 8 0 3; 42 9 0 3; 43 10 0 3; 44 4 0 4; 45 0 0 4; 46 1 0 4; 47 2 0 4; 48 3 0 4; 49 4 0 4; 50 5 0 4; 51 6 0 4; 52 7 0 4; 53 8 0 4; 54 9 0 4; 55 10 0 4; 56 0 0 5; 57 1 0 5; 58 2 0 5; 59 3 0 5; 60 4 0 5; 61 5 0 5; 62 6 0 5; 63 7 0 5; 64 8 0 5; 65 9 0 5; 66 10 0 5; 67 0 0 6; 68 1 0 6; 69 2 0 6; 70 3 0 6; 71 4 0 6; 72 5 0 6; 73 6 0 6; 74 7 0 6; 75 8 0 6; 76 9 0 6; 77 10 0 6; 78 0 0 7; 79 1 0 7; 80 2 0 7; 81 3 0 7; 82 4 0 7; 83 5 0 7; 84 6 0 7; 85 7 0 7; 86 8 0 7; 87 9 0 7; 88 10 0 7; 89 0 0 8; 90 1 0 8; 91 2 0 8; 92 3 0 8; 93 4 0 8; 94 5 0 8; 95 6 0 8; 96 7 0 8; 97 8 0 8; 98 9 0 8; 99 10 0 8; 100 0 0 9; 101 1 0 9; 102 2 0 9; 103 3 0 9; 104 4 0 9; 105 5 0 9; 106 6 0 9; 107 7 0 9; 108 8 0 9; 109 9 0 9; 110 10 0 9; 111 0 0 10; 112 1 0 10; 113 2 0 10; 114 3 0 10; 115 4 0 10; 116 5 0 10; 117 6 0 10; 118 7 0 10; 119 8 0 10; 120 9 0 10; 121 10 0 10;

ELEMENT INCIDENCES SHELL
1 1 2 12; 2 2 3 13; 3 3 4 14; 4 4 5 15; 5 5 6 16; 6 6 7 17; 7 7 8 18; 8 8 9 19; 9 9 10 20; 10 10 11 21; 11 12 2 13; 12 13 3 14; 13 14 4 15; 14 15 5 16; 15 16 6 17; 16 17 7 18; 17 18 8 19; 18 19 9 20; 19 20 10 21; 20 21 11 22; 21 12 13 23; 22 13 14 24; 23 14 15 25; 24 15 16 26; 25 16 17 27; 26 17 18 28; 27 18 19 29; 28 19 20 30; 29 20 21 31;
ELEMENT PROPERTY
1 TO 200 THICKNESS 5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.4
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED BUT FY
2 TO 10 FIXED BUT FX FY MZ
12 23 34 45 56 67 78 89 100 FIXED BUT FY FZ MX
11 22 33 44 55 66 77 88 99 110 FIXED BUT MZ
111 TO 120 FIXED BUT MX
121 FIXED
LOAD 1 UNIFORM PRESSURE
ELEMENT LOAD
1 TO 200 PR -1000
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1 9 73

Verification Examples
V.04 Plate and Shell Elements

STAAD.Pro
3360
User Manual
V. 2D Tapered Beam In-Plane Stress

To find element stress due to joint load at the fixed end of a tapered plate with one end fixed.

Reference


Problem

The tapered plate structure is loaded at the free end. Calculate the maximum stress at the midspan.

\[
E = 30,000.0 \text{ ksi}
\]

Thickness = 2 in

Poisson’s ratio = 0.2
The STAAD.Pro result is taken as average of stress in elements 9 and 11 at node 16 = 0.5(8,333.35 + 8,359.62) = 8,346.5.

Table 415: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum stress at the center (psi)</td>
<td>8,333</td>
<td>8,346.5</td>
<td>0.2%</td>
</tr>
</tbody>
</table>

**STAAD Input**

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

```
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Tapered Beam In-Plane Stress.STD is typically installed with the program.

STAAD SPACE :A TAPERED BEAM WITH PLATE ELEMENTS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*   REFERENCE: CRANDALL, S. H., AND DAHL, N.C., "AN INTRODUCTION TO THE
*           MECHANICS OF SOLIDS," McGRAW-HILL BOOK CO., INC.,
*           NEW YORK, 1978
*
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
  1 50  0  0;  2 50 -4.5  0;  3 50 -9  0;  4 45  0  0;  5 45 -4.2  0;  6 45 -8.4  0;  7 40  0  0;
  8 40 -3.9  0;  9 40 -7.8  0; 10 35  0  0; 11 35 -3.6  0; 12 35 -7.2  0; 13 30  0  0;
  14 30 -3.3  0; 15 30 -6.6  0; 16 25  0  0; 17 25 -3  0; 18 25 -6  0; 19 20  0  0;
```
Verification Examples
V.04 Plate and Shell Elements

STAAD Output

<table>
<thead>
<tr>
<th>ELEMENT STRESSES</th>
<th>FORCE, LENGTH UNITS= POUN INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH</td>
<td></td>
</tr>
<tr>
<td>ELEMENT LOAD SQX SOY MX MY MXY</td>
<td></td>
</tr>
<tr>
<td>TRESCAT TRESCAB SX SY SXY</td>
<td></td>
</tr>
<tr>
<td>9      1         0.00        0.00        0.00        0.00        0.00</td>
<td></td>
</tr>
<tr>
<td>4157.59 4157.59 -4.32 4153.58 -71.59</td>
<td></td>
</tr>
<tr>
<td>4160.36 4160.36</td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 4154.81 SMIN= -5.55 TMAX= 2080.18 ANGLE=-89.0</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 4154.81 SMIN= -5.55 TMAX= 2080.18 ANGLE=-89.0</td>
<td></td>
</tr>
<tr>
<td>JOINT 0.00 0.00 0.00 0.00 0.00</td>
<td></td>
</tr>
<tr>
<td>16 8372.99 8372.99 -78.55 8333.17 38.19</td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 8333.35 SMIN= -78.72 TMAX= 4206.03 ANGLE= 89.7</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 8333.35 SMIN= -78.72 TMAX= 4206.03 ANGLE= 89.7</td>
<td></td>
</tr>
<tr>
<td>JOINT 0.00 0.00 0.00 0.00 0.00</td>
<td></td>
</tr>
<tr>
<td>17 343.34 343.34 42.49 3.59 -196.82</td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 220.82 SMIN= -174.74 TMAX= 197.78 ANGLE=-42.2</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 220.82 SMIN= -174.74 TMAX= 197.78 ANGLE=-42.2</td>
<td></td>
</tr>
<tr>
<td>JOINT 0.00 0.00 0.00 0.00 0.00</td>
<td></td>
</tr>
<tr>
<td>14 295.01 295.01 -107.35 -73.48 -161.24</td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 71.72 SMIN= -252.54 TMAX= 162.13 ANGLE=-48.0</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 71.72 SMIN= -252.54 TMAX= 162.13 ANGLE=-48.0</td>
<td></td>
</tr>
<tr>
<td>JOINT 0.00 0.00 0.00 0.00 0.00</td>
<td></td>
</tr>
<tr>
<td>13 8295.66 8295.66 93.65 8341.38 62.12</td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 8341.85 SMIN= 93.18 TMAX= 4124.34 ANGLE= 89.6</td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= 8341.85 SMIN= 93.18 TMAX= 4124.34 ANGLE= 89.6</td>
<td></td>
</tr>
</tbody>
</table>
V. 2D Surface with Hole Edge Stress

To find the normal stress on the edge of a circular hole in the center of a rectangular plate.

Reference


Problem

Find the normal stress on the edge of the circular hole for the plate shown, when an in-plane load causes tension. Use a one-quarter, doubly symmetric model.
Figure 401: One quarter of rectangular plate with hole

E = 30,000.0 ksi
Size = 12.10 in × 7.0 in
Thickness = 0.1 in
Fillet radius = 1 in
Poisson’s ratio = 0.3
P = 2,000 lbs
Figure 402: Model with nodes and elements labeled

Comparison

Table 416: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress on fillet (node 1, plate 1) (psi)</td>
<td>9.475</td>
<td>9.489</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

STAAD Input

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Surface with Hole Edge Stress.STD is typically installed with the program.

STAAD PLANE :STRESS CONCENTRATION IN A PLATE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* 7. GENERAL CIRCULAR HOLE IN CENTER OF A MEMBER OF RECTANGULAR CROSS SECTION
### Verification Examples

#### V.04 Plate and Shell Elements

| * | 1/4 MODEL DOUBLY SYMMETRIC |
| * | D = 7.0 INCHES WIDTH |
| * | R = 1.0 INCH HOLE RADIUS |
| * | T = 0.1 INCH THICKNESS |
| * | L = 12.1 INCH LENGTH |
| * | LOAD = 2000 POUNDS |

#### THEORETICAL RESULTS

| * | AVERAGE STRESS AT ENDS = 2857 PSI (7 INCH WIDTH) |
| * | AVERAGE NOMINAL STRESS AT CENTER = 4000 PSI (7-5 INCH WIDTH) |
| * | K STRESS CONCENTRATION FACTOR = 2.3688 |
| * | MAXIMUM STRESS AT EDGE OF HOLE = 2.3688 * 4000 = 9475 PSI |

#### STAAD RESULTS

| * | MAXIMUM STRESS AT EDGE OF HOLE = 9489 PSI |
| * | K STRESS CONCENTRATION FACTOR = 2.37225 |
| * | PERCENT ERROR = 0.15% |

#### UNIT INCHES POUND

#### JOINT COORDINATES

| 1 | 0 | 1 | 0; 2 | -0.174 | 0.985 | 0; 3 | -0.342 | 0.94 | 0; 4 | -0.5 | 0.866 | 0; 5 | -0.643 | 0.766 | 0; 6 | -0.766 | 0.643 | 0; 7 | -0.866 | 0.5 | 0; 8 | -0.9 | 0.342 | 0; 9 | -0.985 | 0.174 | 0; 10 | -1 | 0 | 0; 11 | 0 | 1.2 | 0; 12 | -0.208 | 1.182 | 0; 13 | -0.41 | 1.128 | 0.41 | 0; 19 | -1.182 | 0.208 | 0; 20 | -1 | 2 | 0 |

---

**STAAD.Pro** 3367 User Manual

---

**UNIT INCHES POUND**

**JOINT COORDINATES**

| 1 | 0 | 1 | 0; 2 | -0.174 | 0.985 | 0; 3 | -0.342 | 0.94 | 0; 4 | -0.5 | 0.866 | 0; 5 | -0.643 | 0.766 | 0; 6 | -0.766 | 0.643 | 0; 7 | -0.866 | 0.5 | 0; 8 | -0.9 | 0.342 | 0; 9 | -0.985 | 0.174 | 0; 10 | -1 | 0 | 0; 11 | 0 | 1.2 | 0; 12 | -0.208 | 1.182 | 0; 13 | -0.41 | 1.128 | 0.41 | 0; 19 | -1.182 | 0.208 | 0; 20 | -1 | 2 | 0 |

---

**Verification Examples**

#### V.04 Plate and Shell Elements

| 1 | 0 | 1 | 0; 2 | -0.174 | 0.985 | 0; 3 | -0.342 | 0.94 | 0; 4 | -0.5 | 0.866 | 0; 5 | -0.643 | 0.766 | 0; 6 | -0.766 | 0.643 | 0; 7 | -0.866 | 0.5 | 0; 8 | -0.9 | 0.342 | 0; 9 | -0.985 | 0.174 | 0; 10 | -1 | 0 | 0; 11 | 0 | 1.2 | 0; 12 | -0.208 | 1.182 | 0; 13 | -0.41 | 1.128 | 0.41 | 0; 19 | -1.182 | 0.208 | 0; 20 | -1 | 2 | 0 |

---

**STAAD.Pro** 3367 User Manual
## LOT -5.625 2.225 0; 161 -6.05 2.225 0; 162 -1.8 2.65 0;
163 -2.225 2.65 0; 164 -2.65 2.65 0; 165 -3.075 2.65 0; 166 -3.5 2.65 0;
167 -3.925 2.65 0; 168 -4.35 2.65 0; 169 -4.775 2.65 0; 170 -5.2 2.65 0;
171 -5.625 2.65 0; 172 -6.05 2.65 0; 173 -1.8 3.075 0;
174 -2.225 3.075 0; 175 -2.65 3.075 0; 176 -3.075 3.075 0;
177 -3.5 3.075 0; 178 -3.925 3.075 0; 179 -4.35 3.075 0;
180 -4.775 3.075 0; 181 -5.2 3.075 0; 182 -5.625 3.075 0;
183 -6.05 3.075 0; 184 -1.8 3.5 0; 185 -2.225 3.5 0; 186 -2.65 3.5 0;
187 -3.075 3.5 0; 188 -3.5 3.5 0; 189 -3.925 3.5 0; 190 -4.35 3.5 0;
191 -4.775 3.5 0; 192 -5.2 3.5 0; 193 -5.625 3.5 0; 194 -6.05 3.5 0;
201 -1.301 1.301 0;

### ELEMENT INCIDENCES SHELL

<table>
<thead>
<tr>
<th>Element</th>
<th>Incidence</th>
<th>Incidence</th>
<th>Incidence</th>
<th>Incidence</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11</td>
<td>12</td>
<td>2</td>
<td>22</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>12</td>
<td>13</td>
<td>22</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>13</td>
<td>14</td>
<td>23</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>14</td>
<td>15</td>
<td>24</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>15</td>
<td>16</td>
<td>25</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>16</td>
<td>17</td>
<td>26</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>17</td>
<td>18</td>
<td>27</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>18</td>
<td>19</td>
<td>28</td>
</tr>
</tbody>
</table>

### Verification Examples

V.04 Plate and Shell Elements

STAAD.Pro

3368 User Manual
236 67 73 184 173; 237 173 184 185 174; 238 174 185 186 175;
239 175 186 187 176; 240 176 187 188 177; 241 177 188 189 178;
242 178 189 190 179; 243 179 190 191 180; 244 180 191 192 181;
245 181 192 193 182; 246 182 193 194 183;

ELEMENT PROPERTY
1 TO 9 11 TO 19 21 TO 29 31 TO 66 77 TO 86 THICKNESS 0.1
97 TO 106 117 TO 126 137 TO 146 157 TO 166 171 TO 186 191 TO 205 -
206 THICKNESS 0.1
211 TO 226 231 TO 246 THICKNESS 0.1

ELEMENT PLANE STRESS
1 TO 9 11 TO 19 21 TO 29 31 TO 66 77 TO 86 97 TO 106 117 TO 126 137 -
138 TO 146 157 TO 166 171 TO 186 191 TO 206 211 TO 226 231 TO 246

DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL

CONSTANTS
MATERIAL MATERIAL1 ALL

SUPPORTS
10 20 30 40 74 TO 84 FIXED BUT FX
1 11 21 31 44 50 56 62 68 FIXED BUT FY

LOAD 1 TENSILE LOAD

JOINT LOAD
84 FX -42.857
95 106 117 128 139 FX -85.714
150 FX -103.571
161 172 183 FX -121.428
194 FX -60.714

PERFORM ANALYSIS PRINT STATICS CHECK
PRINT JOINT DISPLACEMENTS LIST 6 84
PRINT ELEMENT JOINT STRESSES LIST 1
FINISH

STAAD Output

JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = PLANE
-------------------------------
JOINT LOAD  X-TRANS  Y-TRANS  Z-TRANS  X-ROTAN  Y-ROTAN  Z-ROTAN
6 1 -0.00024 -0.00008 0.00000 0.00000 0.00000 -0.00029
84 1 -0.00066 0.00000 0.00000 0.00000 0.00000 0.00000

*************** END OF LATEST ANALYSIS RESULT ***************

156. PRINT ELEMENT JOINT STRESSES LIST 1

ELEMENT JOINT STRESSES LIST
:STRESS CONCENTRATION IN A PLATE -- PAGE NO.
7

ELEMENT STRESSES  FORCE,LENGTH UNITS= POUN INCH
-----------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

ELEMENT LOAD  SQX  SQY  MX  MY  MXY
VONT  VONB  SX  SY  SXY
TRESCAT  TRESCAB
1 1 0.00 0.00 0.00 0.00 0.00
7461.40 7461.40 650.49 7726.25 -439.78
7753.48 7753.48

TOP : SMAX= 7753.48 SMIN= 623.26 TMAX= 3565.11 ANGLE=-86.5
### V. 2D Circular Surface Edge Stress

The objective of this example is to find the displacement at center, and bending stress at the center and at the perimeter of a circular plate fixed at its periphery.

**Reference**


**Problem**

Find the normal stress on the edge of the circular hole for the plate shown, when an in-plane load causes tension. Use a one-quarter, doubly symmetric model.

\[
\begin{align*}
E &= 10,000.0 \text{ ksi} \\
\text{Radius} &= 10 \text{ in} \\
\text{Thickness} &= 0.02 \text{ in} \\
w &= 0.1 \text{ psi}
\end{align*}
\]
### Verification Examples

V.04 Plate and Shell Elements

#### Comparison

**Table 417: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>δ at node 1 (in)</td>
<td>2.133</td>
<td>2.16005</td>
<td>1.3%</td>
<td>The theoretical results from classical Plate theory was compared with</td>
</tr>
</tbody>
</table>

*Figure 403: Model*
## Verification Examples

### V.04 Plate and Shell Elements

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment at joint 1 (in-lbs)</td>
<td>0.813</td>
<td>0.830</td>
<td>2.6%</td>
<td>the results from Finite Element model - hence the difference in results.</td>
</tr>
</tbody>
</table>

---

### STAAD Input

**Tip**: You can copy and paste this content directly into a .std file to run STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Circular Surface Edge Stress.STD is typically installed with the program.

**STAAD SPACE**: UNIFORM PRESSURE ON A FIXED CIRCULAR PLATE

**START JOB INFORMATION**

**ENGINEER DATE** 14-Sep-18

**END JOB INFORMATION**

* *REFERENCE: "ROARK'S FORMULAS FOR STRESS AND STRAIN", WARREN C. YOUNG, SIXTH EDITION, MCGRAW-HILL, PAGE 429.*

**UNIT INCHES POUND**

**JOINT COORDINATES**

1 0 0 0; 2 1 0 0; 3 0.984808 0 -0.173648; 4 0.939693 0 -0.342019;
5 0.866025 0 -0.5; 6 0.766044 0 -0.642787; 7 0.642787 0 -0.766045;
8 0.5 0 -0.866026; 9 0.34202 0 -0.939693; 10 0.173648 0 -0.984808;
11 -6.39758e-07 0 -1; 12 -0.173649 0 -0.984808;
13 -0.342021 0 -0.939692; 14 -0.500001 0 -0.866025;
15 -0.642788 0 -0.766044; 16 -0.766045 0 -0.642787;
17 -0.866026 0 -0.499999; 18 -0.939693 0 -0.342019;
19 -0.984808 0 -0.173647; 20 -1 0 1.27952e-06; 21 -0.984807 0 0.17365;
22 -0.939692 0 0.342021; 23 -0.866025 0 0.500001;
24 -0.766043 0 0.642789; 25 -0.642786 0 0.766045;
26 -0.499998 0 0.866026; 27 -0.342018 0 0.939693;
28 -0.173646 0 0.984808; 29 1.91927e-06 0 1; 30 0.17365 0 0.984807;
31 0.342022 0 0.939692; 32 0.500002 0 0.866024; 33 0.642789 0 0.766043;
34 0.766046 0 0.642786; 35 0.866027 0 0.499999; 36 0.939694 0 0.342018;
37 0.984808 0 0.173646; 38 1.2916 0 0; 39 1.21371 0 -0.224284;
40 1.21371 0 -0.441753; 41 1.11856 0 -0.6458; 42 0.989423 0 -0.830225;
43 0.830224 0 -0.989423; 44 0.645799 0 -1.1856; 45 0.441753 0 -1.21371;
46 0.224283 0 -1.27198; 47 -8.26311e-07 0 -1.2916;
48 -0.224285 0 -1.27198; 49 -0.441754 0 -1.21371;
50 -0.645801 0 -1.11856; 51 -0.830225 0 -0.989422;
52 -0.989424 0 -0.830224; 53 -1.11856 0 -0.645799;
54 -1.21371 0 -0.441752; 55 -1.27198 0 -0.224282;
56 -1.2916 0 1.65262e-06; 57 -1.27198 0 0.224286;
58 -1.21371 0 0.441755; 59 -1.11856 0 0.645802; 60 -0.989422 0 0.830226;
61 -0.830223 0 0.989424; 62 -0.645798 0 1.11856; 63 -0.441751 0 1.21371;
64 -0.224282 0 1.27198; 65 2.47893e-06 0 1.2916; 66 0.224286 0 1.27198;
67 0.441756 0 1.21371; 68 0.645803 0 1.11856; 69 0.830227 0 0.989421;
70 0.989425 0 0.830222; 71 1.11856 0 0.645797; 72 1.21371 0 0.44175;
73 1.27198 0 0.224281; 74 1.6681 0 0; 75 1.64276 0 -0.289663;
76 1.5675 0 -0.570524; 77 1.44462 0 -0.83405; 78 1.27784 0 -1.07223;
Verification Examples
V.04 Plate and Shell Elements

ELEMENT INCIDENCES SHELL
1 1 2 3; 2 1 3 4; 3 1 4 5; 4 1 5 6; 5 1 6 7; 6 1 7 8; 7 1 8 9; 8 1 9 10; 9 1 10 11; 10 1 11 12; 11 1 12 13; 12 1 13 14; 13 1 14 15; 14 1 15 16; 15 1 16 17; 16 1 17 18; 17 1 18 19; 18 1 19 20; 19 1 20 21; 20 1 21 22; 21 1 22 23; 22 1 23 24; 23 1 24 25; 24 1 25 26; 25 1 26 27; 26 1 27 28; 27 1 28 29; 28 1 29 30; 29 1 30 31; 30 1 31 32; 31 1 32 33; 32 1 33 34;
Verification Examples

V.04 Plate and Shell Elements

STAAD.Pro

3375

User Manual
ELEMENT PROPERTY
1 TO 360 THICKNESS 0.02
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
326 TO 361 FIXED
LOAD 1 UNIFORM PRESSURE
ELEMENT LOAD
1 TO 360 PR 0.1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1
PRINT ELEMENT JOINT STRESSES LIST 1
PRINT SUPPORT REACTION LIST 326
FINISH

STAAD Output

**JOINT DISPLACEMENT (INCH RADIANS) STRUCTURE TYPE = SPACE**

---

**JOINT LOAD**

<table>
<thead>
<tr>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>2.1605</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

277. PRINT ELEMENT JOINT STRESSES LIST 1

**ELEMENT JOINT STRESSES LIST**

:UNIFORM PRESSURE ON A FIXED CIRCULAR PLATE -- PAGE NO.

8

**ELEMENT STRESSES**

<table>
<thead>
<tr>
<th>FORCE,LENGTH UNITS= POUN INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT</td>
<td>VONB</td>
<td>SX</td>
<td>SY</td>
<td>SXY</td>
<td></td>
</tr>
<tr>
<td>TRESCAT</td>
<td>TRESCAB</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-1.48</td>
<td>-2.18</td>
<td>0.83</td>
<td>0.83</td>
<td>0.00</td>
</tr>
<tr>
<td>12399.08</td>
<td>12399.08</td>
<td>12399.08</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 12399.08 SMIN= -12399.08 TMAX= 0.00 ANGLE= 90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= -12399.07 SMIN= 12399.08 TMAX= 0.00 ANGLE= 90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>-1.48</td>
<td>-2.18</td>
<td>0.85</td>
<td>0.83</td>
<td>0.00</td>
</tr>
<tr>
<td>12566.42</td>
<td>12566.42</td>
<td>12566.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 12566.42 SMIN= -12566.42 TMAX= 0.00 ANGLE= 90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= -12436.87 SMIN= 12436.87 TMAX= 0.00 ANGLE= -90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>-1.48</td>
<td>-2.18</td>
<td>0.82</td>
<td>0.83</td>
<td>0.00</td>
</tr>
<tr>
<td>12344.11</td>
<td>12344.11</td>
<td>12344.11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 12344.87 SMIN= -12344.87 TMAX= 0.00 ANGLE= 90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= -12249.21 SMIN= 12249.21 TMAX= 0.00 ANGLE= -90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>-1.48</td>
<td>-2.18</td>
<td>0.82</td>
<td>0.82</td>
<td>0.00</td>
</tr>
<tr>
<td>12289.85</td>
<td>12289.85</td>
<td>12289.85</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX= 12289.85 SMIN= -12289.85 TMAX= 0.00 ANGLE= 90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX= -12255.94 SMIN= 12255.94 TMAX= 0.00 ANGLE= -90.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>-1.48</td>
<td>-2.18</td>
<td>0.82</td>
<td>0.82</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

MAXIMUM PRINCIPAL STRESS = 1.269208E+04
MINIMUM PRINCIPAL STRESS = -1.269208E+04
MAXIMUM SHEAR STRESS = 1.276060E+02
MAXIMUM VONMISES STRESS = 1.239908E+04
MAXIMUM TRESCA STRESS = 1.239908E+04

PLATE NO. 1
CASE NO. 1

*************** END OF ELEMENT FORCES ***************

278. PRINT SUPPORT REACTION LIST 326

**SUPPORT REACTION LIST 326**

:UNIFORM PRESSURE ON A FIXED CIRCULAR PLATE -- PAGE NO.

9

**SUPPORT REACTIONS**

-UNIT POUN INCH STRUCTURE TYPE = SPACE

---

**Verification Examples**

V.04 Plate and Shell Elements

**STAAD.Pro 3377 User Manual**
V. 2D Retaining Wall

To find the bending moments at various points on a wall fixed along three edges.

Reference

Problem
Find the bending moment at the points circled in the figure for 2 load cases:

a. a uniform pressure over the entire wall
b. a hydrostatic pressure varying linearly from 0 at the top to maximum at the bottom.

\[
E = 3,150 \text{ ksi} \\
\text{Length, } a = 60 \text{ ft} \\
\text{Height, } b = 40 \text{ ft}
\]

*Figure 404: Model*
**Hand Calculation**

\[
\frac{a}{b} = \frac{60\text{ft}}{40\text{ft}} = 1.5
\]

\[
M = \frac{\beta_i q b^2}{6}
\]

**a.** For uniform pressure (load case 1):

where

\[
\beta_i = \text{value taken for Case 10a, Table 26, on p.469 of the reference (when } a/b = 1.5):
\]

- 0.727 for midpoint of bottom edge (moment about the horizontal axis)
- 0.484 for mid point of top edge (free edge; moment about the horizontal axis)
- 1.073 for corner of the free edge and fixed edge (moment about the vertical axis)

\[
q = \text{magnitude of the pressure, 1 ksf}
\]

**b.** For linearly varying pressure (load case 2):

Pressure varies from zero intensity at the top to maximum at the base.

where

\[
\beta_i = \text{value taken for Case 10d, Table 26, on p.470 of the reference (when } a/b = 1.5):
\]

- 0.351 for midpoint of bottom edge (moment about the horizontal axis)
- 0.244 for corner of the free edge and fixed edge (moment about the vertical axis)

\[
q = \text{magnitude of the pressure at bottom, 3.5 ksf}
\]

**Comparison**

**Table 418: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uniform pressure load case</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bending moment at midpoint of</td>
<td>193.9</td>
<td>393.25/2= 196.6</td>
<td>1.4%</td>
<td>The theoretical results from classical Plate theory was compared with the results from Finite Element model - hence the difference in results.</td>
</tr>
<tr>
<td>bottom edge (ft·kip/ft)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bending moment at midpoint of</td>
<td>129</td>
<td>130.6</td>
<td>1.2%</td>
<td></td>
</tr>
<tr>
<td>top edge (ft·kip/ft)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

**V.04 Plate and Shell Elements**

#### Results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending moment at corner of free edge and fixed edge (ft·kip/ft)</td>
<td>286.1</td>
<td>294.0</td>
<td>2.8%</td>
<td></td>
</tr>
<tr>
<td>Linearly varying pressure load case</td>
<td>327.6</td>
<td>673.0/2 = 336.5</td>
<td>2.7%</td>
<td></td>
</tr>
<tr>
<td>Bending moment at corner of free edge and fixed edge (ft·kip/ft)</td>
<td>227.7</td>
<td>212.0</td>
<td>6.9%</td>
<td></td>
</tr>
</tbody>
</table>

#### STAAD Input

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Retaining Wall.STD is typically installed with the program.

```
STAAD SPACE : A WALL FIXED ALONG 2 EDGES
START JOB INFORMATION
ENGINEER DATE 21-Aug-18
END JOB INFORMATION
*
* REFERENCE 'ROARK'S FORMULAS FOR STRESS AND STRAIN', WARREN YOUNG, 6TH ED.
* CASES 10A & 10D, PP.469-470
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 40 0; 3 30 0 0; 4 30 40 0; 5 60 0 0; 6 60 40 0; 7 0 2 0; 8 0 4 0; 9 0 6 0; 10 0 8 0; 11 0 10 0; 12 0 12 0; 13 0 14 0; 14 0 16 0; 15 0 18 0; 16 0 20 0; 17 0 22 0; 18 0 24 0; 19 0 26 0; 20 0 28 0; 21 0 30 0; 22 0 32 0; 23 0 34 0; 24 0 36 0; 25 0 38 0; 26 2 0 0; 27 2 2 0; 28 2 4 0; 29 2 6 0; 30 2 8 0; 31 2 10 0; 32 2 12 0; 33 2 14 0; 34 2 16 0; 35 2 18 0; 36 2 20 0; 37 2 22 0; 38 2 24 0; 39 2 26 0; 40 2 28 0; 41 2 30 0; 42 2 32 0; 43 2 34 0; 44 2 36 0; 45 2 38 0; 46 2 40 0; 47 4 0 0; 48 4 2 0; 49 4 4 0; 50 4 6 0; 51 4 8 0; 52 4 10 0; 53 4 12 0; 54 4 14 0; 55 4 16 0; 56 4 18 0; 57 4 20 0; 58 4 22 0; 59 4 24 0; 60 4 26 0; 61 4 28 0; 62 4 30 0; 63 4 32 0; 64 4 34 0; 65 4 36 0; 66 4 38 0; 67 4 40 0; 68 6 0 0; 69 6 2 0; 70 6 4 0; 71 6 6 0; 72 6 8 0; 73 6 10 0; 74 6 12 0; 75 6 14 0; 76 6 16 0; 77 6 18 0; 78 6 20 0; 79 6 22 0; 80 6 24 0; 81 6 26 0; 82 6 28 0; 83 6 30 0;
```
### Verification Examples

**STAAD.Pro**

<table>
<thead>
<tr>
<th>Example</th>
<th>Part Numbers</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11 12 32 31</td>
</tr>
<tr>
<td>2</td>
<td>10 15 16 36 35</td>
</tr>
<tr>
<td>3</td>
<td>14 19 20 38 35</td>
</tr>
<tr>
<td>4</td>
<td>18 23 24 44 43</td>
</tr>
<tr>
<td>5</td>
<td>22 27 28 49 48</td>
</tr>
<tr>
<td>6</td>
<td>26 31 32 33 54 53</td>
</tr>
</tbody>
</table>
Verification Examples

V.04 Plate and Shell Elements

STAAD.Pro

3383

User Manual
Verification Examples
V.04 Plate and Shell Elements

STAAD.Pro User Manual
Verification Examples

V.04 Plate and Shell Elements

STAAD.Pro 3385 User Manual
Verification Examples
V.04 Plate and Shell Elements
Verification Examples
V.04 Plate and Shell Elements

STAAD Output

SUPPORT REACTIONS -UNIT KIP FEET  STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 1</td>
<td>0.00</td>
<td>0.00</td>
<td>24.50</td>
<td>-59.94</td>
<td>-293.96</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>-3.73</td>
<td>-42.71</td>
<td>-211.97</td>
<td>0.00</td>
</tr>
<tr>
<td>3 1</td>
<td>0.00</td>
<td>0.00</td>
<td>53.00</td>
<td>393.27</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>112.98</td>
<td>673.01</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

************* END OF LATEST ANALYSIS RESULT *************

389. PRINT ELEMENT JOINT STRESSES LIST 300

ELEMENT JOINT STRESSES LIST
: A WALL FIXED ALONG 2 EDGES

-- PAGE NO.

10

ELEMENT STRESSES  FORCE,LENGTH UNITS= KIP FEET

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
<th>TRESCAT</th>
<th>TRESCAB</th>
</tr>
</thead>
<tbody>
<tr>
<td>300 1</td>
<td>-0.88</td>
<td>1.09</td>
<td>2.76</td>
<td>126.17</td>
<td>-2.01</td>
<td>187.30</td>
<td>187.30</td>
</tr>
<tr>
<td></td>
<td>189.31</td>
<td>189.31</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>189.31</td>
<td>4.09</td>
<td>92.61</td>
<td>ANGLE=-89.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>-4.09</td>
<td>-189.31</td>
<td>92.61</td>
<td>ANGLE= 0.9</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>-0.92</td>
<td>0.26</td>
<td>4.26</td>
<td>125.10</td>
<td>-2.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>318</td>
<td>184.65</td>
<td>184.65</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>187.73</td>
<td>6.32</td>
<td>90.70</td>
<td>ANGLE=-88.8</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>-6.32</td>
<td>-187.73</td>
<td>90.70</td>
<td>ANGLE= 1.2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>-0.92</td>
<td>1.92</td>
<td>0.87</td>
<td>124.07</td>
<td>-1.27</td>
<td></td>
<td></td>
</tr>
<tr>
<td>319</td>
<td>185.49</td>
<td>185.49</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>186.13</td>
<td>1.29</td>
<td>92.42</td>
<td>ANGLE=-89.4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX=</td>
<td>-1.29</td>
<td>-186.13</td>
<td>92.42</td>
<td>ANGLE= 0.6</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>-0.84</td>
<td>1.92</td>
<td>1.43</td>
<td>130.57</td>
<td>-1.57</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>194.83</td>
<td>194.83</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP : SMAX=</td>
<td>195.88</td>
<td>2.11</td>
<td>96.88</td>
<td>ANGLE=-89.3</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

**V.04 Plate and Shell Elements**

#### V.04 Plate and Shell Elements

| BOTT: SMAX= | -2.11 | SMIN= | -195.88 | TMAX= | 96.88 | ANGLE= | 0.7 |
| Joint | -0.84 | 0.26 | 4.47 | 124.94 | -2.76 |
| 338 | 184.30 | 184.30 | 0.00 | 0.00 | 0.00 |
| TOP : SMAX= | 187.51 | SMIN= | 6.61 | TMAX= | 90.45 | ANGLE= | -88.7 |
| Joint | -6.61 | 116.85 | 1.3 |
| 2 | -0.92 | 0.78 | 2.26 | 116.85 | -1.13 |
| 173.62 | 173.62 | 0.00 | 0.00 | 0.00 |
| TOP : SMAX= | 187.51 | SMIN= | 6.61 | TMAX= | 90.45 | ANGLE= | -88.7 |
| Joint | -6.61 | 116.85 | 1.3 |
| 2 | -0.92 | 0.78 | 2.26 | 116.85 | -1.13 |
| 173.62 | 173.62 | 0.00 | 0.00 | 0.00 |
| TOP : SMAX= | 187.51 | SMIN= | 6.61 | TMAX= | 90.45 | ANGLE= | -88.7 |
| Joint | -6.61 | 116.85 | 1.3 |
| 2 | -0.92 | 0.78 | 2.26 | 116.85 | -1.13 |
| 173.62 | 173.62 | 0.00 | 0.00 | 0.00 |
| TOP : SMAX= | 187.51 | SMIN= | 6.61 | TMAX= | 90.45 | ANGLE= | -88.7 |
| Joint | -6.61 | 116.85 | 1.3 |
| 2 | -0.92 | 0.78 | 2.26 | 116.85 | -1.13 |
| 173.62 | 173.62 | 0.00 | 0.00 | 0.00 |

**11**

| MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES **** |
| PRINCIPAL | PRINCIPAL | SHEAR | VONMISES | TRESCA |
| PLATE NO. | 300 | 300 | 300 | 300 | 300 |
| CASE NO. | 1 | 1 | 1 | 1 | 1 |

**V. Thermal Load on a Plate**

Find deflections and moments due to thermal loading and compare theoretical answers to the STAAD solution.

**Reference**


**Problem**

A rectangular plate is simply supported on all four sides. The transverse and longitudinal bending moments as well as the deflections at several points on the plate are computed.
W = 12 in., W = 16 in.

The plate is modeled using 1 in. X 1 in. size elements. At the corner nodes, all the degrees of freedom are considered restrained. For the nodes along the four edges, rotation is permitted about that edge.

**Theoretical Solution**

From the Reference, equation (j), the expression for deflection normal to the plate surface is:

\[
w = -\frac{\alpha t}{\pi^3 h} \sum_{m=1,3,5,...}^{\infty} \sin \frac{\max \alpha}{m^3} \left( 1 - \frac{\cosh \frac{m\gamma}{a}}{\cosh \alpha_m} \right)
\]

where

\[
\alpha_m = \frac{mnb}{2a}
\]

From the Reference, equation (k), the expressions for bending moment per unit width are

\[
M_x = \frac{4D\alpha t(1 - \nu^2)}{nh} \sum_{m=1,3,5,...}^{\infty} \sin \frac{\max \alpha}{m^3} \frac{\cosh \frac{m\gamma}{a}}{m \cdot \cosh \alpha_m}
\]

\[
M_y = \frac{\alpha t(1 - \nu^2)D}{h} - \frac{4D\alpha t(1 - \nu^2)}{nh} \sum_{m=1,3,5,...}^{\infty} \sin \frac{\max \alpha}{m^3} \frac{\cosh \frac{m\gamma}{a}}{m \cdot \cosh \alpha_m}
\]

where

\[
\alpha = \text{Coefficient of Thermal Expansion}
\]

\[
t = \text{Difference between the temperatures of the upper and lower surfaces of the plate}
\]

\[
\nu = \text{Poisson's ratio}
\]

\[
h = \text{Plate thickness}
\]

\[
a = \text{Dimension of the plate along the x1 axis}
\]

\[
b = \text{Dimension of the plate along the x2 axis}
\]

\[
E = \text{Elastic Modulus}
\]

\[
D = \frac{Eh^3}{12(1 - \nu^2)}
\]
The numerical values used for this example are:

\[
\begin{align*}
\alpha &= 12.0E-06 / \degree F \\
t &= 450 \degree F \\
\nu &= 0.3 \\
h &= 0.3 \text{ in.} \\
a &= 12 \text{ in.} \\
b &= 16 \text{ in.} \\
E &= 10.0E6 \text{ psi}
\end{align*}
\]

Comparison

Table 419: Comparison of results

<table>
<thead>
<tr>
<th>Node Number</th>
<th>X</th>
<th>Y</th>
<th>Theoretical Deflection</th>
<th>STAAD.ProDeflection</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>6</td>
<td>-8</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>6</td>
<td>-7</td>
<td>0.0897</td>
<td>0.0897</td>
</tr>
<tr>
<td>33</td>
<td>6</td>
<td>-6</td>
<td>0.1597</td>
<td>0.1597</td>
</tr>
<tr>
<td>46</td>
<td>6</td>
<td>-5</td>
<td>0.2132</td>
<td>0.2132</td>
</tr>
<tr>
<td>59</td>
<td>6</td>
<td>-4</td>
<td>0.2531</td>
<td>0.2531</td>
</tr>
<tr>
<td>72</td>
<td>6</td>
<td>-3</td>
<td>0.2818</td>
<td>0.2818</td>
</tr>
<tr>
<td>85</td>
<td>6</td>
<td>-2</td>
<td>0.3011</td>
<td>0.3011</td>
</tr>
<tr>
<td>98</td>
<td>6</td>
<td>-1</td>
<td>0.3122</td>
<td>0.3122</td>
</tr>
<tr>
<td>111</td>
<td>6</td>
<td>0</td>
<td>0.3158</td>
<td>0.3158</td>
</tr>
<tr>
<td>124</td>
<td>6</td>
<td>1</td>
<td>0.3122</td>
<td>0.3122</td>
</tr>
</tbody>
</table>

Table 420: Comparison of results

<table>
<thead>
<tr>
<th>Node Number</th>
<th>X</th>
<th>Y</th>
<th>Theoretical Deflection</th>
<th>STAAD.ProDeflection</th>
</tr>
</thead>
<tbody>
<tr>
<td>118</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>119</td>
<td>1</td>
<td>1</td>
<td>0.1004</td>
<td>0.1004</td>
</tr>
<tr>
<td>120</td>
<td>2</td>
<td>1</td>
<td>0.1794</td>
<td>0.1794</td>
</tr>
<tr>
<td>121</td>
<td>3</td>
<td>1</td>
<td>0.2387</td>
<td>0.2387</td>
</tr>
<tr>
<td>122</td>
<td>4</td>
<td>1</td>
<td>0.2799</td>
<td>0.2799</td>
</tr>
</tbody>
</table>
### Table 421: Comparison of results

<table>
<thead>
<tr>
<th>Element Number</th>
<th>X</th>
<th>Y</th>
<th>Theoretical Moment (Pound-in/in)</th>
<th>STAAD STAAD.Pro Moment (Pound-in/in)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Mx</td>
<td>My</td>
</tr>
<tr>
<td>97</td>
<td>0.5</td>
<td>0.5</td>
<td>16.74</td>
<td>388.26</td>
</tr>
<tr>
<td>98</td>
<td>1.5</td>
<td>0.5</td>
<td>48.93</td>
<td>356.07</td>
</tr>
<tr>
<td>99</td>
<td>2.5</td>
<td>0.5</td>
<td>77.45</td>
<td>327.55</td>
</tr>
<tr>
<td>100</td>
<td>3.5</td>
<td>0.5</td>
<td>100.36</td>
<td>304.64</td>
</tr>
<tr>
<td>101</td>
<td>4.5</td>
<td>0.5</td>
<td>116.31</td>
<td>288.69</td>
</tr>
<tr>
<td>102</td>
<td>5.5</td>
<td>0.5</td>
<td>124.47</td>
<td>280.54</td>
</tr>
<tr>
<td>103</td>
<td>6.5</td>
<td>0.5</td>
<td>124.47</td>
<td>280.54</td>
</tr>
<tr>
<td>104</td>
<td>7.5</td>
<td>0.5</td>
<td>116.31</td>
<td>288.69</td>
</tr>
<tr>
<td>105</td>
<td>8.5</td>
<td>0.5</td>
<td>100.36</td>
<td>304.64</td>
</tr>
<tr>
<td>106</td>
<td>9.5</td>
<td>0.5</td>
<td>77.45</td>
<td>327.55</td>
</tr>
<tr>
<td>107</td>
<td>10.5</td>
<td>0.5</td>
<td>48.93</td>
<td>356.07</td>
</tr>
<tr>
<td>108</td>
<td>11.5</td>
<td>0.5</td>
<td>16.74</td>
<td>388.26</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Node Number</th>
<th>X</th>
<th>Y</th>
<th>Theoretical Deflection</th>
<th>STAAD.ProDeflection</th>
</tr>
</thead>
<tbody>
<tr>
<td>123</td>
<td>5</td>
<td>1</td>
<td>0.3042</td>
<td>0.3042</td>
</tr>
<tr>
<td>124</td>
<td>6</td>
<td>1</td>
<td>0.3122</td>
<td>0.3122</td>
</tr>
<tr>
<td>125</td>
<td>7</td>
<td>1</td>
<td>0.3042</td>
<td>0.3042</td>
</tr>
<tr>
<td>126</td>
<td>8</td>
<td>1</td>
<td>0.2799</td>
<td>0.2799</td>
</tr>
<tr>
<td>127</td>
<td>9</td>
<td>1</td>
<td>0.2387</td>
<td>0.2387</td>
</tr>
<tr>
<td>128</td>
<td>10</td>
<td>1</td>
<td>0.1794</td>
<td>0.1794</td>
</tr>
<tr>
<td>129</td>
<td>11</td>
<td>1</td>
<td>0.1004</td>
<td>0.1004</td>
</tr>
<tr>
<td>130</td>
<td>12</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Verification Examples
V.04 Plate and Shell Elements
### Table 422: Comparison of results

<table>
<thead>
<tr>
<th>Element Number</th>
<th>X</th>
<th>Y</th>
<th>Theoretical Moment (Pound-in/in)</th>
<th>STAAD.ProMoment (Pound-in/in)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Mx</td>
<td>My</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>5.5</td>
<td>-7.5</td>
<td>373.88</td>
<td>31.12</td>
</tr>
<tr>
<td>18</td>
<td>5.5</td>
<td>-6.5</td>
<td>311.64</td>
<td>93.36</td>
</tr>
<tr>
<td>30</td>
<td>5.5</td>
<td>-5.5</td>
<td>256.83</td>
<td>148.17</td>
</tr>
<tr>
<td>42</td>
<td>5.5</td>
<td>-4.5</td>
<td>211.14</td>
<td>193.87</td>
</tr>
<tr>
<td>54</td>
<td>5.5</td>
<td>-3.5</td>
<td>175.41</td>
<td>229.60</td>
</tr>
<tr>
<td>66</td>
<td>5.5</td>
<td>-2.5</td>
<td>149.47</td>
<td>255.53</td>
</tr>
<tr>
<td>78</td>
<td>5.5</td>
<td>-1.5</td>
<td>132.68</td>
<td>272.32</td>
</tr>
<tr>
<td>90</td>
<td>5.5</td>
<td>-0.5</td>
<td>124.47</td>
<td>280.54</td>
</tr>
<tr>
<td>102</td>
<td>5.5</td>
<td>0.5</td>
<td>124.47</td>
<td>280.54</td>
</tr>
<tr>
<td>114</td>
<td>5.5</td>
<td>1.5</td>
<td>132.68</td>
<td>272.32</td>
</tr>
<tr>
<td>126</td>
<td>5.5</td>
<td>2.5</td>
<td>149.47</td>
<td>255.53</td>
</tr>
<tr>
<td>138</td>
<td>5.5</td>
<td>3.5</td>
<td>175.41</td>
<td>229.59</td>
</tr>
<tr>
<td>150</td>
<td>5.5</td>
<td>4.5</td>
<td>211.14</td>
<td>193.87</td>
</tr>
<tr>
<td>162</td>
<td>5.5</td>
<td>5.5</td>
<td>256.83</td>
<td>148.17</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Thermal Load on a Plate.STD is typically installed with the program.

**STAAD SPACE**

START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* Thermal loading on a simply supported rectangular plate

UNIT INCHES POUND

JOINT COORDINATES
1 0 -8 0; 2 1 -8 0; 3 2 -8 0; 4 3 -8 0; 5 4 -8 0; 6 5 -8 0; 7 6 -8 0; 8 7 -8 0; 9 8 -8 0; 10 9 -8 0; 11 10 -8 0; 12 11 -8 0; 13 12 -8 0; 14 0 -7 0; 15 1 -7 0; 16 2 -7 0; 17 3 -7 0; 18 4 -7 0; 19 5 -7 0; 20 6 -7 0; 21 7 -7 0; 22 8 -7 0; 23 9 -7 0; 24 10 -7 0; 25 11 -7 0; 26 12 -7 0; 27 0 -6 0; 28 1 -6 0; 29 2 -6 0; 30 3 -6 0; 31 4 -6 0; 32 5 -6 0; 33 6 -6 0; 34 7 -6 0; 35 8 -6 0; 36 9 -6 0; 37 10 -6 0;
### Verification Examples

#### V.04 Plate and Shell Elements

<table>
<thead>
<tr>
<th>ELEMENT INCIDENCES SHELL</th>
<th>218 9 8 0</th>
<th>219 10 8 0</th>
<th>220 11 8 0</th>
<th>221 12 8 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>212 3 8 0</td>
<td>213 4 8 0</td>
<td>214 5 8 0</td>
<td>215 6 8 0</td>
<td>216 7 8 0</td>
</tr>
<tr>
<td>206 10 7 0</td>
<td>207 11 7 0</td>
<td>208 12 7 0</td>
<td>209 0 8 0</td>
<td>210 1 8 0</td>
</tr>
<tr>
<td>212 3 8 0</td>
<td>213 4 8 0</td>
<td>214 5 8 0</td>
<td>215 6 8 0</td>
<td>216 7 8 0</td>
</tr>
<tr>
<td>218 9 8 0</td>
<td>219 10 8 0</td>
<td>220 11 8 0</td>
<td>221 12 8 0</td>
<td></td>
</tr>
</tbody>
</table>
| ELEMENT INCIDENCES SHELL | 1 1 2 15 14; 2 2 3 16 15; 3 3 4 17 16; 4 4 5 18 17; 5 5 6 19 18; 6 6 7 20 19; 7 7 8 21 20; 8 8 9 22 21; 9 9 10 23 22; 10 10 11 24 23; 11 11 12 25 24; 12 12 13 26 25; 13 13 14 27 26; 14 14 15 28 27; 15 15 16 29 28; 16 16 17 30 29; 17 17 18 31 30; 18 18 19 32 31; 19 19 20 33 32; 20 20 21 34 33; 21 21 22 35 34; 22 22 23 36 35; 23 23 24 37 36; 24 24 25 38 37; 25 25 26 39 38; 26 26 27 40 39; 27 27 28 41 40; 28 28 29 42 41; 29 29 30 43 42; 30 30 31 44 43; 31 31 32 45 44; 32 32 33 46 45; 33 33 34 47 46; 34 34 35 48 47; 35 35 36 49 48; 36 36 37 50 49; 37 37 38 51 50; 38 38 39 52 51; 39 39 40 53 52; 40 40 41 54 53; 41 41 42 55 54; 42 42 43 56 55; 43 43 44 57 56; 44 44 45 58 57; 45 45 46 59 58; 46 46 47 60 59; 47 47 48 61 60; 48 48 49 62 61; 49 49 50 63 62; 50 50 51 64 63; 51 51 52 65 64; 52 52 53 66 65; 53 53 54 67 66; 54 54 55 68 67; 55 55 56 69 68; 56 56 57 70 69; 57 57 58 71 70; 58 58 59 72 71; 59 59 60 73 72; 60 60 61 74 73; 61 61 62 75 74; 62 62 63 76 75; 63 63 64 77 76; 64 64 65 78 77; 65 65 66 79 78; 66 66 67 80 79; 67 67 68 81 80; 68 68 69 82 81; 69 69 70 83 82; 70 70 71 84 83; 71 71 72 85 84; 72 72 73 86 85; 73 73 74 87 86; 74 74 75 88 87; 75 75 76 89 88; 76 76 77 90 89; 77 77 78 91 90; 78 78 79 92 91; 79 79 80 93 92; 80 80 81 94 93; 81 81 82 95 94; 82 82 83 96 95; 83 83 84 97 96; 84 84 85 98 97; 85 85 86 99 98; 86 86 87 100 99; 87 87 88 101 100; 88 88 89 102 101; 89 89 90 103 102; 90 90 91 104 103; 91 91 92 105 104; 92 92 93 106 105; 93 93 94 107 106; 94 94 95 108 107; 95 95 96 109 108; 96 96 97 110 109; 97 97 98 111 110; 98 98 99 112 111; 99 99 100 113 112; 100 100 101 114 113; 101 101 102 115 114; 102 102 103 116 115; 103 103 104 117 116; 104 104 105 118 117; 105 105 106 119 118; 106 106 107 120 119; 107 107 108 121 120; 108 108 109 122 121; 109 109 110 123 122; 110 110 111 124 123;
ELEMENT PROPERTY
1 TO 192 THICKNESS 0.3
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
ALPHA 1.2e-05
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 13 209 221 FIXED
2 TO 12 210 TO 220 FIXED BUT MX
14 26 27 39 40 52 53 65 66 78 79 91 92 104 105 117 118 130 131 143 -
144 156 157 169 170 182 183 195 196 208 FIXED BUT MY
LOAD 1
TEMPERATURE LOAD
1 TO 192 TEMP 0 450
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 7 20 33 46 59 72 85 98 111 118 TO 130 -
137 150 163 176 189 202 215
PRINT ELEMENT STRESSES LIST 6 18 30 42 54 66 78 90 97 TO 108 114 126 -
138 150 162 174 186
FINISH
### STAAD Output

#### JOINT DISPLACEMENT (INCH RADIANS)  
**STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.10098</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.08974</td>
<td>0.07940</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>33</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.15969</td>
<td>0.06132</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>46</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.21318</td>
<td>0.04637</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>59</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.25310</td>
<td>0.03406</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>72</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.28181</td>
<td>0.02380</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>85</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.30109</td>
<td>0.01506</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>98</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.31218</td>
<td>0.00729</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>111</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.31579</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>118</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.11185</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>119</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.10040</td>
<td>-0.00193</td>
<td>-0.08944</td>
<td>0.00000</td>
</tr>
<tr>
<td>120</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.17937</td>
<td>-0.00371</td>
<td>-0.06893</td>
<td>0.00000</td>
</tr>
<tr>
<td>121</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.23869</td>
<td>-0.00521</td>
<td>-0.05010</td>
<td>0.00000</td>
</tr>
<tr>
<td>122</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.27991</td>
<td>-0.00635</td>
<td>-0.03262</td>
<td>0.00000</td>
</tr>
<tr>
<td>123</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.30417</td>
<td>-0.00705</td>
<td>-0.01607</td>
<td>0.00000</td>
</tr>
<tr>
<td>124</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.31218</td>
<td>-0.00729</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>125</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.30417</td>
<td>-0.00705</td>
<td>0.01607</td>
<td>0.00000</td>
</tr>
<tr>
<td>126</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.27991</td>
<td>-0.00635</td>
<td>0.03262</td>
<td>0.00000</td>
</tr>
<tr>
<td>127</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.23869</td>
<td>-0.00521</td>
<td>0.05010</td>
<td>0.00000</td>
</tr>
<tr>
<td>128</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.17937</td>
<td>-0.00371</td>
<td>0.06893</td>
<td>0.00000</td>
</tr>
<tr>
<td>129</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.10040</td>
<td>-0.00193</td>
<td>0.08944</td>
<td>0.00000</td>
</tr>
<tr>
<td>130</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.11185</td>
<td>0.00000</td>
</tr>
<tr>
<td>131</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.39109</td>
<td>-0.01506</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>132</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.28181</td>
<td>-0.02380</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>133</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.25310</td>
<td>-0.03406</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>134</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.21318</td>
<td>-0.04637</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>135</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.15969</td>
<td>-0.06132</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>136</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.08974</td>
<td>-0.07940</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>137</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.10098</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

123. PRINT ELEMENT STRESSES LIST 6 18 30 42 54 66 78 90 97 TO 108 114 126 -
ELEMENT STRESSES LIST 6
124. 138 150 162 174 186

--- PAGE NO. 5
---

### ELEMENT STRESSES

**FORCE,LENGTH UNITS= POUN INCH**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>1</td>
<td>0.34</td>
<td>4.36</td>
<td>-373.27</td>
<td>-32.17</td>
<td>-31.51</td>
</tr>
<tr>
<td></td>
<td></td>
<td>24160.11</td>
<td>24160.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>25076.87</td>
<td>25076.87</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :  SMAX= -1951.93  SMIN= -25076.87  TMAX= 11562.47  ANGLE=-84.8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX=  25076.87  SMIN=  1951.93  TMAX=  11562.47  ANGLE= 5.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>0.74</td>
<td>2.75</td>
<td>-312.19</td>
<td>-93.95</td>
<td>-29.19</td>
</tr>
<tr>
<td></td>
<td></td>
<td>18798.88</td>
<td>18798.88</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>21068.45</td>
<td>21068.45</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP :  SMAX= -6007.83  SMIN= -21068.45  TMAX=  7530.31  ANGLE=-82.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT: SMAX=  21068.45  SMIN=  6007.83  TMAX=  7530.31  ANGLE= 7.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.04 Plate and Shell Elements

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT/VONB</td>
<td></td>
<td>SX</td>
<td>SY</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TRECT/VRECT</td>
<td></td>
<td>SX</td>
<td>SY</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### STAAD SPACE

**Element Stresses**

<table>
<thead>
<tr>
<th>Element</th>
<th>Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>97</td>
<td>1</td>
</tr>
<tr>
<td>98</td>
<td>1</td>
</tr>
<tr>
<td>99</td>
<td>1</td>
</tr>
<tr>
<td>100</td>
<td>1</td>
</tr>
</tbody>
</table>

**Stress Calculation**

- **Stress** = Force/Unit Width/Thickness, **Moment** = Force-Length/Unit Width

**Force, Length Units:** POUN INCH

---

**STAAD Pro User Manual**

---

**Verification Examples**

- **V.04 Plate and Shell Elements**

---

**STAAD SPACE**

---

**Page No.**

---
<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>101</td>
<td>1</td>
<td>-0.42</td>
<td>-0.16</td>
<td>-116.93</td>
<td>-289.32</td>
<td>6.05</td>
</tr>
<tr>
<td></td>
<td></td>
<td>16820.56</td>
<td>16820.56</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>13902.28</td>
<td>13902.28</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -7780.89</td>
<td>SMIN= -19302.28</td>
<td>TMAX= 5760.70</td>
<td>ANGLE= 2.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 19302.28</td>
<td>SMIN= 7780.89</td>
<td>TMAX= 5760.70</td>
<td>ANGLE= -88.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>102</td>
<td>1</td>
<td>-0.13</td>
<td>-0.13</td>
<td>-125.11</td>
<td>-281.19</td>
<td>2.05</td>
</tr>
<tr>
<td></td>
<td></td>
<td>16269.07</td>
<td>16269.07</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>18747.84</td>
<td>18747.84</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -8338.78</td>
<td>SMIN= -18747.84</td>
<td>TMAX= 5204.53</td>
<td>ANGLE= -0.8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 18747.84</td>
<td>SMIN= 8338.78</td>
<td>TMAX= 5204.53</td>
<td>ANGLE= 89.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>103</td>
<td>1</td>
<td>0.13</td>
<td>-0.13</td>
<td>-125.11</td>
<td>-281.19</td>
<td>-2.05</td>
</tr>
<tr>
<td></td>
<td></td>
<td>16269.07</td>
<td>16269.07</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>18747.84</td>
<td>18747.84</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -8338.78</td>
<td>SMIN= -18747.84</td>
<td>TMAX= 5204.53</td>
<td>ANGLE= -0.8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 18747.84</td>
<td>SMIN= 8338.78</td>
<td>TMAX= 5204.53</td>
<td>ANGLE= 89.2</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

STAAD SPACE
-- PAGE NO.

7

ELEMENT STRESSES
FORCE,LENGTH UNITS= POUN INCH
----------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>VONT</th>
<th>VONB</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>105</td>
<td>1</td>
<td>0.78</td>
<td>-0.21</td>
<td>-100.93</td>
<td>-305.20</td>
<td>-9.76</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>17989.87</td>
<td>17989.87</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>20377.92</td>
<td>20377.92</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -6697.43</td>
<td>SMIN= -20377.92 TMAX= 6840.24 ANGLE= -2.7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 20377.92</td>
<td>SMIN= 6697.43 TMAX= 6840.24 ANGLE= 87.3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>106</td>
<td>1</td>
<td>1.23</td>
<td>-0.23</td>
<td>-77.92</td>
<td>-328.01</td>
<td>-12.92</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>19844.15</td>
<td>19844.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>21911.50</td>
<td>21911.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -5150.16</td>
<td>SMIN= -21911.50 TMAX= 8380.67 ANGLE= -3.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 21911.50</td>
<td>SMIN= 5150.16 TMAX= 8380.67 ANGLE= 87.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>107</td>
<td>1</td>
<td>1.69</td>
<td>-0.20</td>
<td>-49.25</td>
<td>-356.37</td>
<td>-15.27</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>22367.84</td>
<td>22367.84</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>23808.31</td>
<td>23808.31</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -3232.73</td>
<td>SMIN= -23808.31 TMAX= 10287.79 ANGLE= -2.8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 23808.31</td>
<td>SMIN= 3232.73 TMAX= 10287.79 ANGLE= 87.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>108</td>
<td>1</td>
<td>2.00</td>
<td>-0.08</td>
<td>-16.85</td>
<td>-388.37</td>
<td>-16.53</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>25419.74</td>
<td>25419.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>25939.96</td>
<td>25939.96</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -1074.52</td>
<td>SMIN= -25939.96 TMAX= 12432.72 ANGLE= -2.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 25939.96</td>
<td>SMIN= 1074.52 TMAX= 12432.72 ANGLE= 87.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>114</td>
<td>1</td>
<td>-0.09</td>
<td>-0.36</td>
<td>-133.36</td>
<td>-273.00</td>
<td>6.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>15779.24</td>
<td>15779.24</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>18218.24</td>
<td>18218.24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -8871.89</td>
<td>SMIN= -18218.24 TMAX= 4673.17 ANGLE= 2.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 18218.24</td>
<td>SMIN= 8871.89 TMAX= 4673.17 ANGLE= -87.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>126</td>
<td>1</td>
<td>-0.02</td>
<td>-0.49</td>
<td>-150.19</td>
<td>-256.25</td>
<td>10.66</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>14918.57</td>
<td>14918.57</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>17154.23</td>
<td>17154.23</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOP</td>
<td>SMAX= -9942.06</td>
<td>SMIN= -17154.23 TMAX= 3606.09 ANGLE= 5.7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOTT:</td>
<td>SMAX= 17154.23</td>
<td>SMIN= 9942.06 TMAX= 3606.09 ANGLE= -84.3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Verification Examples
V.04 Plate and Shell Elements

STAAD.Pro
3397
User Manual
### V.05 Solids

#### V. Cantilever Beam End Displacement 1

To find the displacement at the free end of a cantilever beam modeled with solid elements.

**Reference**

Hand calculation.

---

**STAAD Space**

---

**STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH**

---

**Verification Examples**

V.05 Solids

---

**3398**

User Manual
Problem

Calculate the maximum displacement of a cantilever beam due to a concentrated load at the free end.

$L = 10$ in
$A = 2$ in$^2$
$P = 300$ lb
\[ l = \frac{2}{3} \text{ in.} \]
\[ E = 29,000 \text{ ksi} \]
\[ \nu = 0.3 \]

**Hand Calculation**

\[ \delta_{\text{bend}} = \frac{PL}{3EI} = \frac{300(10)}{3[29(10)^6](2/3)} = 0.00517 \text{ in} \]
\[ \delta_{\text{shear}} = \frac{12/5*(1+\nu)PL}{AE} = \frac{12/5*(1+0.3)(300)(10)}{[29(10)^6(2)]} = 0.00016 \text{ in} \]
\[ \delta = \delta_{\text{bend}} + \delta_{\text{shear}} = 0.00517 + 0.00016 = 0.00533 \text{ in} \]

**Comparison**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection, ( \delta ) (in)</td>
<td>0.00533</td>
<td>0.00529</td>
<td>&lt; 1.0%</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\05 Solids\Cantilever Beam End Displacement 1.STD is typically installed with the program.

**STAAD SPACE : A CANTILEVER BEAM WITH SOLID ELEMENTS**

**START JOB INFORMATION**

**ENGINEER DATE 17-Sep-18**

**END JOB INFORMATION**

**INPUT WIDTH 72**

**UNIT INCHES POUND**

**JOINT COORDINATES**

1 0 0 0; 2 0.5 0 0; 3 1 0 0; 4 1.5 0 0; 5 2 0 0; 6 2.5 0 0; 7 3 0 0;
8 3.5 0 0; 9 4 0 0; 10 4.5 0 0; 11 5 0 0; 12 5.5 0 0; 13 6 0 0;
14 6.5 0 0; 15 7 0 0; 16 7.5 0 0; 17 8 0 0; 18 8.5 0 0; 19 9 0 0;
20 9.5 0 0; 21 10 0 0; 22 0 0 0.25; 23 0.5 0 0.25; 24 1 0 0.25;
25 1.5 0 0.25; 26 2 0 0.25; 27 2.5 0 0.25; 28 3 0 0.25; 29 3.5 0 0.25;
30 4 0 0.25; 31 4.5 0 0.25; 32 5 0 0.25; 33 5.5 0 0.25; 34 6 0 0.25;
35 6.5 0 0.25; 36 7 0 0.25; 37 7.5 0 0.25; 38 8 0 0.25; 39 8.5 0 0.25;
40 9 0 0.25; 41 9.5 0 0.25; 42 10 0 0.25; 43 0 0 0.5; 44 0.5 0 0.5;
45 1 0 0.5; 46 1.5 0 0.5; 47 2 0 0.5; 48 2.5 0 0.5; 49 3 0 0.5;
50 3.5 0 0.5; 51 4 0 0.5; 52 4.5 0 0.5; 53 5 0 0.5; 54 5.5 0 0.5;
55 6 0 0.5; 56 6.5 0 0.5; 57 7 0 0.5; 58 7.5 0 0.5; 59 8 0 0.5;
60 8.5 0 0.5; 61 9 0 0.5; 62 9.5 0 0.5; 63 10 0 0.5; 64 0 0 0.75;
65 0.5 0 0.75; 66 1 0 0.75; 67 1.5 0 0.75; 68 2 0 0.75; 69 2.5 0 0.75;
70 3 0 0.75; 71 3.5 0 0.75; 72 4 0 0.75; 73 4.5 0 0.75; 74 5 0 0.75;
75 5.5 0 0.75; 76 6 0 0.75; 77 6.5 0 0.75; 78 7 0 0.75; 79 7.5 0 0.75;
80 8 0 0.75; 81 8.5 0 0.75; 82 9 0 0.75; 83 9.5 0 0.75; 84 10 0 0.75;
85 0 0 1; 86 0.5 0 1; 87 1 0 1; 88 1.5 0 1; 89 2 0 1; 90 2.5 0 1;
91 3 0 1; 92 3.5 0 1; 93 4 0 1; 94 4.5 0 1; 95 5 0 1; 96 5.5 0 1;
97 6 0 1; 98 6.5 0 1; 99 7 0 1; 100 7.5 0 1; 101 8 0 1; 102 8.5 0 1;
103 9 0 1; 104 9.5 0 1; 105 10 0 1; 106 0 0.5 0; 107 0.5 0 0;
108 1 0.5 0; 109 1.5 0.5 0; 110 2 0.5 0; 111 2.5 0.5 0; 112 3 0.5 0;
### Verification Examples

#### V.05 Solids

| STAAD.Pro | 371 | 6.5 | 1.5 | 0.5 | 372 | 7 | 1.5 | 0.5 | 373 | 7.5 | 1.5 | 0.5 | 374 | 8 | 1.5 | 0.5 |
|-----------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
| 367       | 4.5 | 1.5 | 0.5 | 368 | 5 | 1.5 | 0.5 | 369 | 5.5 | 1.5 | 0.5 | 370 | 6 | 1.5 | 0.5 |
| 363       | 2.5 | 1.5 | 0.5 | 364 | 3 | 1.5 | 0.5 | 365 | 3.5 | 1.5 | 0.5 | 366 | 4 | 1.5 | 0.5 |
| 351       | 7   | 1.5 | 0.25 | 352 | 7.5 | 1.5 | 0.25 | 353 | 8 | 1.5 | 0.25 | 354 | 8.5 | 1.5 | 0.25 |
| 347       | 5   | 1.5 | 0.25 | 348 | 5.5 | 1.5 | 0.25 | 349 | 6 | 1.5 | 0.25 | 350 | 6.5 | 1.5 | 0.25 |
| 335       | 9.5 | 1 | 1 | 336 | 10 | 1.5 | 0.25 | 337 | 0 | 1.5 | 0.25 | 338 | 0.5 | 1.5 | 0.25 |
| 325       | 4.5 | 1 | 0 | 326 | 5 | 1 | 0 | 327 | 5.5 | 1 | 0 | 328 | 6 | 1.5 | 0 |
| 320       | 2 | 1.5 | 0 | 321 | 2.5 | 1.5 | 0 | 322 | 3 | 1.5 | 0 | 323 | 3.5 | 1.5 | 0 |
| 315       | 10 | 1 | 0 | 316 | 0 | 1.5 | 0 | 317 | 1 | 1 | 0 | 318 | 1.5 | 1 | 0 |
| 314       | 9.5 | 1.5 | 0.25 | 315 | 10 | 0.5 | 0.75 | 316 | 0 | 1.5 | 0.25 | 317 | 1 | 1.5 | 0.75 |
| 313       | 9 | 0.5 | 0.5 | 314 | 9.5 | 1 | 0.25 | 315 | 10 | 0.5 | 0.75 | 316 | 0 | 1.5 | 0.25 |
| 312       | 8.5 | 1 | 0.25 | 313 | 9 | 0.5 | 0.5 | 314 | 9.5 | 1 | 0.25 | 315 | 10 | 0.5 | 0.75 |
| 311       | 8 | 0.5 | 0.5 | 312 | 8.5 | 1 | 0.25 | 313 | 9 | 0.5 | 0.5 | 314 | 9.5 | 1 | 0.25 |
| 310       | 7.5 | 1 | 0.25 | 311 | 8 | 0.5 | 0.5 | 312 | 8.5 | 1 | 0.25 | 313 | 9 | 0.5 | 0.5 |
| 309       | 7 | 1 | 0.25 | 310 | 7.5 | 1 | 0.25 | 311 | 8 | 0.5 | 0.5 | 312 | 8.5 | 1 | 0.25 |
| 308       | 6.5 | 1 | 0.25 | 309 | 7 | 1 | 0.25 | 310 | 7.5 | 1 | 0.25 | 311 | 8 | 0.5 | 0.5 |
| 307       | 6 | 1 | 0.25 | 308 | 6.5 | 1 | 0.25 | 309 | 7 | 1 | 0.25 | 310 | 7.5 | 1 | 0.25 |
| 306       | 5.5 | 1 | 0.25 | 307 | 6 | 1 | 0.25 | 308 | 6.5 | 1 | 0.25 | 309 | 7 | 1 | 0.25 |
| 305       | 5 | 1 | 0.25 | 306 | 5.5 | 1 | 0.25 | 307 | 6 | 1 | 0.25 | 308 | 6.5 | 1 | 0.25 |
| 304       | 4.5 | 1 | 0.25 | 305 | 5 | 1 | 0.25 | 306 | 5.5 | 1 | 0.25 | 307 | 6 | 1 | 0.25 |
| 303       | 4 | 1 | 0.25 | 304 | 4.5 | 1 | 0.25 | 305 | 5 | 1 | 0.25 | 306 | 5.5 | 1 | 0.25 |
| 302       | 3 | 1 | 0.25 | 303 | 4 | 1 | 0.25 | 304 | 4.5 | 1 | 0.25 | 305 | 5 | 1 | 0.25 |
| 301       | 2 | 1 | 0.25 | 302 | 3 | 1 | 0.25 | 303 | 4 | 1 | 0.25 | 304 | 4.5 | 1 | 0.25 |
| 300       | 1 | 1 | 0.25 | 301 | 2 | 1 | 0.25 | 302 | 3 | 1 | 0.25 | 303 | 4 | 1 | 0.25 |
| 299       | 1 | 1 | 0 | 300 | 1 | 1 | 0 | 301 | 2 | 1 | 0 | 302 | 3 | 1 | 0 |
| 298       | 0.5 | 1 | 0 | 299 | 1 | 1 | 0 | 300 | 1 | 1 | 0 | 301 | 2 | 1 | 0 |
| 297       | 1 | 0 | 0 | 298 | 0.5 | 1 | 0 | 299 | 1 | 1 | 0 | 300 | 1 | 1 | 0 |
| 296       | 0.5 | 0.5 | 0 | 297 | 1 | 0 | 0 | 298 | 0.5 | 1 | 0 | 299 | 1 | 1 | 0 |

### STAAD.Pro

3401

User Manual
<table>
<thead>
<tr>
<th>ELEMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>INCIDENCES</td>
</tr>
<tr>
<td>SOLID</td>
</tr>
<tr>
<td>520</td>
</tr>
<tr>
<td>521</td>
</tr>
<tr>
<td>522</td>
</tr>
<tr>
<td>523</td>
</tr>
<tr>
<td>524</td>
</tr>
<tr>
<td>525</td>
</tr>
<tr>
<td>514</td>
</tr>
<tr>
<td>515</td>
</tr>
<tr>
<td>516</td>
</tr>
<tr>
<td>517</td>
</tr>
<tr>
<td>518</td>
</tr>
<tr>
<td>519</td>
</tr>
<tr>
<td>503</td>
</tr>
<tr>
<td>504</td>
</tr>
<tr>
<td>505</td>
</tr>
<tr>
<td>506</td>
</tr>
<tr>
<td>507</td>
</tr>
<tr>
<td>508</td>
</tr>
<tr>
<td>509</td>
</tr>
<tr>
<td>510</td>
</tr>
<tr>
<td>511</td>
</tr>
<tr>
<td>512</td>
</tr>
<tr>
<td>513</td>
</tr>
<tr>
<td>514</td>
</tr>
<tr>
<td>515</td>
</tr>
<tr>
<td>516</td>
</tr>
<tr>
<td>517</td>
</tr>
<tr>
<td>518</td>
</tr>
<tr>
<td>519</td>
</tr>
<tr>
<td>520</td>
</tr>
<tr>
<td>521</td>
</tr>
<tr>
<td>522</td>
</tr>
<tr>
<td>523</td>
</tr>
<tr>
<td>524</td>
</tr>
<tr>
<td>525</td>
</tr>
</tbody>
</table>

V.05 Solids

Verification Examples

STAAD.Pro
Verification Examples
V.05 Solids
Verification Examples
V.05 Solids

| 250 346 451 430 325 347 452 431 326; |
| 251 347 452 431 326 348 453 432 327; |
| 252 348 453 432 327 349 454 433 328; |
| 253 349 454 433 328 350 455 434 329; |
| 254 350 455 434 329 351 456 435 330; |
| 255 351 456 435 330 352 457 436 331; |
| 256 352 457 436 331 353 458 437 332; |
| 257 353 458 437 332 354 459 438 333; |
| 258 354 459 438 333 355 460 439 334; |
| 259 355 460 439 334 356 461 440 335; |
| 260 356 461 440 335 357 462 441 336; |
| 261 358 463 442 337 359 464 443 338; |
| 262 359 464 443 338 360 465 444 339; |
| 263 360 465 444 339 361 466 445 340; |
| 264 361 466 445 340 362 467 446 341; |
| 265 362 467 446 341 363 468 447 342; |
| 266 363 468 447 342 364 469 448 343; |
| 267 364 469 448 343 365 470 449 344; |
| 268 365 470 449 344 366 471 450 345; |
| 269 366 471 450 345 367 472 451 346; |
| 270 367 472 451 346 368 473 452 347; |
| 271 368 473 452 347 369 474 453 348; |
| 272 369 474 453 348 370 475 454 349; |
| 273 370 475 454 349 371 476 455 350; |
| 274 371 476 455 350 372 477 456 351; |
| 275 372 477 456 351 373 478 457 352; |
| 276 373 478 457 352 374 479 458 353; |
| 277 374 479 458 353 375 480 459 354; |
| 278 375 480 459 354 376 481 460 355; |
| 279 376 481 460 355 377 482 461 356; |
| 280 377 482 461 356 378 483 462 357; |
| 281 379 484 463 358 380 485 464 359; |
| 282 380 485 464 359 381 486 465 360; |
| 283 381 486 465 360 382 487 466 361; |
| 284 382 487 466 361 383 488 467 362; |
| 285 383 488 467 362 384 489 468 363; |
| 286 384 489 468 363 385 490 469 364; |
| 287 385 490 469 364 386 491 470 365; |
| 288 386 491 470 365 387 492 471 366; |
| 289 387 492 471 366 388 493 472 367; |
| 290 388 493 472 367 389 494 473 368; |
| 291 389 494 473 368 390 495 474 369; |
| 292 390 495 474 369 391 496 475 370; |
| 293 391 496 475 370 392 497 476 371; |
| 294 392 497 476 371 393 498 477 372; |
| 295 393 498 477 372 394 499 478 373; |
| 296 394 499 478 373 395 500 479 374; |
| 297 395 500 479 374 396 501 480 375; |
| 298 396 501 480 375 397 502 481 376; |
| 299 397 502 481 376 398 503 482 377; |
| 300 398 503 482 377 399 504 483 378; |
| 301 400 505 484 399 506 485 380; |
| 302 401 506 485 380 402 507 486 381; |
| 303 402 507 486 381 403 508 487 382; |
| 304 403 508 487 382 404 509 488 383; |
| 305 404 509 488 383 405 510 489 384; |
| 306 405 510 489 384 406 511 490 385; |
| 307 406 511 490 385 407 512 491 386; |
V. Cantilever Beam End Displacement 2

To find the displacement at the free end and normal stresses at mid-span of a cantilever beam modeled with solid elements.
Hand Calculation

Displacement due to Load 1:

\[ \delta_{L1} = \frac{PL}{AE} = \frac{1,200(15)}{10 \times (10)^6} = 0.0003 \text{ in} \]

Rotate due to Load 2:

\[ \phi_{L2} = TL/(c_2ab^3G) \]

where

\[ a = \text{long side of the cross section} = 3 \text{ in} \]
\[ b = \text{short side of the cross section} = 2 \text{ in} \]
\[ c_2 = 0.1958 \text{ for } a/b = 1.5 \]
\[ G = E/[2(1+\nu)] = 10(10)^3/(2(1+0.3)) = 3,846 \text{ ksi} \]

\[ \phi_{L2} = 2000(15)/(0.1958(3)(2)^33.846(10)^6) = 0.00166 \text{ rad} \]

Displacement due to Load 3:

\[ \delta_{L3} = ML^2/(2EI) = 2500(15)^2/[2(10)(10)^7(4.5)] = 0.00625 \text{ in} \]

Displacement due to Load 4:

\[ \delta_{bend} = PL^3/(3EI) = 1000(15)^3/[3(10(10)^6)(4.5)] = 0.025 \text{ in} \]
\[ \delta_{shear} = 12/5*(1+\nu)PL/AE = 12/5*(1+0.3)(1000)(15)/[(6)(10)(10)^6] = 0.00078 \text{ in} \]
\[ \delta_{L4} = \delta_{bend} + \delta_{shear} = 0.025 + 0.00078 = 0.02578 \text{ in} \]

Stress at midspan due to Load 1:

\[ \sigma_a = P/A = 1200/6 = 200 \text{ psi} \]

Stress at midspan due to Load 3:

\[ \sigma_b = My/I = 2500(1.5)/4.5 = 833.33 \text{ psi} \]

Stress at midspan due to Load 4:

\[ \sigma_b = My/I = 7.5(1000)(1.5)/4.5 = 2,500 \text{ psi} \]

Comparison

Table 424: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Displacement, ( \delta ) (in)</td>
<td>LC1</td>
<td>0.00030</td>
<td>0.00033</td>
<td>10% The theoretical results from classical Beam theory was compared with the results from the model with Solid Elements - hence the</td>
</tr>
<tr>
<td></td>
<td>LC3</td>
<td>0.00625</td>
<td>0.00621</td>
<td>0.6%</td>
</tr>
<tr>
<td></td>
<td>LC4</td>
<td>0.02578</td>
<td>0.02537</td>
<td>1.6%</td>
</tr>
<tr>
<td>Maximum Rotation, ( \phi ) (rad)</td>
<td>LC2</td>
<td>0.00166</td>
<td>0.00149</td>
<td>10.2%</td>
</tr>
</tbody>
</table>
### Result Type

<table>
<thead>
<tr>
<th>Normal Stress at Midspan* (psi)</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>LC1</td>
<td>200.0</td>
<td>200.004</td>
<td>none</td>
<td>difference in results.</td>
</tr>
<tr>
<td>LC3</td>
<td>833.3</td>
<td>833.325</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>LC4</td>
<td>2,500</td>
<td>2,630.409</td>
<td>5.2%</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** (*) Stresses computed at Node no. 32 of solid no. 10.

### STAAD Input

The file C:\\Users\\Public\\Documents\\STAAD.Pro CONNECT Edition\\Samples\\Verification Models\\05 Solids\\Cantilever Beam End Displacement 2.STD is typically installed with the program.

**STAAD SPACE : A CANTILEVER BEAM WITH SOLID ELEMENTS**

**START JOB INFORMATION**

**ENGINEER DATE** 17-Sep-18

**END JOB INFORMATION**

**INPUT WIDTH** 72

**UNIT INCHES POUND**

**JOINT COORDINATES**

1 0 0 0; 2 0.75 0 0; 3 1.5 0 0; 4 2.25 0 0; 5 3 0 0; 6 3.75 0 0;
7 4.5 0 0; 8 5.25 0 0; 9 6 0 0; 10 6.75 0 0; 11 7.5 0 0; 12 8.25 0 0;
13 9 0 0; 14 9.75 0 0; 15 10.5 0 0; 16 11.25 0 0; 17 12 0 0;
18 12.75 0 0; 19 13.5 0 0; 20 14.25 0 0; 21 15 0 0; 22 0 0 0.5;
23 0.75 0 0.5; 24 1.5 0 0.5; 25 2.25 0 0.5; 26 3 0 0.5; 27 3.75 0 0.5;
28 4.5 0 0.5; 29 5.25 0 0.5; 30 6 0 0.5; 31 6.75 0 0.5; 32 7.5 0 0.5;
33 8.25 0 0.5; 34 9 0 0.5; 35 9.75 0 0.5; 36 10.5 0 0.5; 37 11.25 0 0.5;
38 12 0 0.5; 39 12.75 0 0.5; 40 13.5 0 0.5; 41 14.25 0 0.5; 42 15 0 0.5;
43 0 0 1; 44 0.75 0 1; 45 1.5 0 1; 46 2.25 0 1; 47 3 0 1; 48 3.75 0 1;
49 4.5 0 1; 50 5.25 0 1; 51 6 0 1; 52 6.75 0 1; 53 7.5 0 1; 54 8.25 0 1;
55 9 0 1; 56 9.75 0 1; 57 10.5 0 1; 58 11.25 0 1; 59 12 0 1;
60 12.75 0 1; 61 13.5 0 1; 62 14.25 0 1; 63 15 0 1; 64 0 0 1.5;
65 0.75 0 1.5; 66 1.5 0 1.5; 67 2.25 0 1.5; 68 3 0 1.5; 69 3.75 0 1.5;
70 4.5 0 1.5; 71 5.25 0 1.5; 72 6 0 1.5; 73 6.75 0 1.5; 74 7.5 0 1.5;
75 8.25 0 1.5; 76 9 0 1.5; 77 9.75 0 1.5; 78 10.5 0 1.5; 79 11.25 0 1.5;
80 12 0 1.5; 81 12.75 0 1.5; 82 13.5 0 1.5; 83 14.25 0 1.5; 84 15 0 1.5;
85 0 0 2; 86 0.75 0 2; 87 1.5 0 2; 88 2.25 0 2; 89 3 0 2; 90 3.75 0 2;
91 4.5 0 2; 92 5.25 0 2; 93 6 0 2; 94 6.75 0 2; 95 7.5 0 2; 96 8.25 0 2;
97 9 0 2; 98 9.75 0 2; 99 10.5 0 2; 100 11.25 0 2; 101 12 0 2;
102 12.75 0 2; 103 13.5 0 2; 104 14.25 0 2; 105 15 0 2; 106 0 0 0.75 0;
107 0.75 0 0.75 0; 108 1.5 0.75 0; 109 2.25 0.75 0; 110 3 0.75 0;
111 3.75 0 0.75 0; 112 4.5 0.75 0; 113 5.25 0.75 0; 114 6 0.75 0;
115 6.75 0.75 0; 116 7.5 0.75 0; 117 8.25 0.75 0; 118 9 0.75 0;
119 9.75 0.75 0; 120 10.5 0.75 0; 121 11.25 0.75 0; 122 12 0.75 0;
123 12.75 0.75 0; 124 13.5 0.75 0; 125 14.25 0.75 0; 126 15 0.75 0;
127 0 0.75 0.5; 128 0.75 0.75 0.5; 129 1.5 0.75 0.5; 130 2.25 0.75 0.5;
131 3 0.75 0.5; 132 3.75 0.75 0.5; 133 4.5 0.75 0.5; 134 5.25 0.75 0.5;
135 6 0.75 0.5; 136 6.75 0.75 0.5; 137 7.5 0.75 0.5; 138 8.25 0.75 0.5;
139 9 0.75 0.5; 140 9.75 0.75 0.5; 141 10.5 0.75 0.5;
142 11.25 0.75 0.5; 143 12 0.75 0.5; 144 12.75 0.75 0.5;
145 13.5 0.75 0.5; 146 14.25 0.75 0.5; 147 15 0.75 0.5; 148 0 0.75 1;
149 0.75 0.75 1; 150 1.5 0.75 1; 151 2.25 0.75 1; 152 3 0.75 1;
Verification Examples

V.05 Solids

STAAD.Pro 3410 User Manual
SINCE SOLID-ONLY MODELS HAVE NO ROTATIONAL DOF AT THEIR NODES, ADD DUMMY MEMBERS TO CREATE THOSE DEGREES OF FREEDOM.

MEMBER INCIDENCES
1001 525 273; 1002 105 273; 1003 21 273; 1004 441 273;

ELEMENT INCIDENCES SOLID
1 22 127 106 1 23 128 107; 2 23 128 107 2 24 129 108; 3 24 129 108 3 25 130 110; 4 25 130 109 4 26 131 110; 5 25 130 110 5 26 131 110 5 27 132 111; 6 27 132 111 6 28 133 112; 7 28 133 112 7 29 134 113; 8 28 133 112 8 29 134 113 8 30 135 114; 9 30 135 114 9 31 136 115; 10 31 136 115 10 32 137 116; 11 32 137 116 11 33 138 117; 12 33 138 117 12 34 139 118; 13 34 139 118 13 35 140 119; 14 35 140 119 14 36 141 120; 15 36 141 120 15 37 142 121; 16 37 142 121 16 38 143 122; 17 38 143 122 17 39 144 123; 18 38 143 122 18 39 144 123 18 40 145 124; 19 40 145 124 19 41 146 125; 20 41 146 125 20 42 147 126; 21 42 147 126 21 43 148 127; 22 43 148 127 22 44 149 128; 23 44 149 128 23 45 150 129; 24 45 150 129 24 46 151 130; 25 46 151 130 25 47 152 131; 26 47 152 131 26 48 153 132; 27 48 153 132 27 49 154 133; 28 49 154 133 28 50 155 134; 29 50 155 134 29 51 156 135; 30 51 156 135 30 52 157 136; 31 52 157 136 31 53 158 137; 32 53 158 137 32 54 159 138; 33 54 159 138 33 55 160 139; 34 55 160 139 34 56 161 140; 35 56 161 140 35 57 162 141; 36 57 162 141 36 58 163 142; 37 58 163 142 37 59 164 143; 38 59 164 143 38;
Verification Examples

V.05 Solids
Verification Examples

V.05 Solids

127 175 280 259 154 176 281 260 155;
128 176 281 260 155 177 282 261 156;
129 177 282 261 156 178 283 262 157;
130 178 283 262 157 179 284 263 158;
131 179 284 263 158 180 285 264 159;
132 180 285 264 159 181 286 265 160;
133 181 286 265 160 182 287 266 161;
134 182 287 266 161 183 288 267 162;
135 183 288 267 162 184 289 268 163;
136 184 289 268 163 185 290 269 164;
137 185 290 269 164 186 291 270 165;
138 186 291 270 165 187 292 271 166;
139 187 292 271 166 188 293 272 167;
140 188 293 272 167 189 294 273 168;
141 190 295 274 169 191 296 275 170;
142 191 296 275 170 192 297 276 171;
143 192 297 276 171 193 298 277 172;
144 193 298 277 172 194 299 278 173;
145 194 299 278 173 195 300 279 174;
146 195 300 279 174 196 301 280 175;
147 196 301 280 175 197 302 281 176;
148 197 302 281 176 198 303 282 177;
149 198 303 282 177 199 304 283 178;
150 199 304 283 178 200 305 284 179;
151 200 305 284 179 201 306 285 180;
152 201 306 285 180 202 307 286 181;
153 202 307 286 181 203 308 287 182;
154 203 308 287 182 204 309 288 183;
155 204 309 288 183 205 310 289 184;
156 205 310 289 184 206 311 290 185;
157 206 311 290 185 207 312 291 186;
158 207 312 291 186 208 313 292 187;
159 208 313 292 187 209 314 293 188;
160 209 314 293 188 210 315 294 189;
161 232 337 316 211 233 338 317 212;
162 233 338 317 212 234 339 318 213;
163 234 339 318 213 235 340 319 214;
164 235 340 319 214 236 341 320 215;
165 236 341 320 215 237 342 321 216;
166 237 342 321 216 238 343 322 217;
167 238 343 322 217 239 344 323 218;
168 239 344 323 218 240 345 324 219;
169 240 345 324 219 241 346 325 220;
170 241 346 325 220 242 347 326 221;
171 242 347 326 221 243 348 327 222;
172 243 348 327 222 244 349 328 223;
173 244 349 328 223 245 350 329 224;
174 245 350 329 224 246 351 330 225;
175 246 351 330 225 247 352 331 226;
176 247 352 331 226 248 353 332 227;
177 248 353 332 227 249 354 333 228;
178 249 354 333 228 250 355 334 229;
179 250 355 334 229 251 356 335 230;
180 251 356 335 230 252 357 336 231;
181 252 357 336 231 253 358 337 232;
182 253 358 337 232 254 359 338 233;
183 255 360 339 234 256 361 340 235;
184 256 361 340 235 257 362 341 236;
### Verification Examples

#### V.05 Solids

| 185 257 362 341 236 258 363 342 237; |
| 186 258 363 342 237 259 364 343 238; |
| 187 259 364 343 238 260 365 344 239; |
| 188 260 365 344 239 261 366 345 240; |
| 189 261 366 345 240 262 367 346 241; |
| 190 262 367 346 241 263 368 347 242; |
| 191 263 368 347 242 264 369 348 243; |
| 192 264 369 348 243 265 370 349 244; |
| 193 265 370 349 244 266 371 350 245; |
| 194 266 371 350 245 267 372 351 246; |
| 195 267 372 351 246 268 373 352 247; |
| 196 268 373 352 247 269 374 353 248; |
| 197 269 374 353 248 270 375 354 249; |
| 198 270 375 354 249 271 376 355 250; |
| 199 271 376 355 250 272 377 356 251; |
| 200 272 377 356 251 273 378 357 252; |
| 201 274 379 358 253 275 380 359 254; |
| 202 275 380 359 254 276 381 360 255; |
| 203 276 381 360 255 277 382 361 256; |
| 204 277 382 361 256 278 383 362 257; |
| 205 278 383 362 257 279 384 363 258; |
| 206 279 384 363 258 280 385 364 259; |
| 207 280 385 364 259 281 386 365 260; |
| 208 281 386 365 260 282 387 366 261; |
| 209 282 387 366 261 283 388 367 262; |
| 210 283 388 367 262 284 389 368 263; |
| 211 284 389 368 263 285 390 369 264; |
| 212 285 390 369 264 286 391 370 265; |
| 213 286 391 370 265 287 392 371 266; |
| 214 287 392 371 266 288 393 372 267; |
| 215 288 393 372 267 289 394 373 268; |
| 216 289 394 373 268 289 395 374 269; |
| 217 290 395 374 269 291 396 375 270; |
| 218 291 396 375 270 292 397 376 271; |
| 219 292 397 376 271 293 398 377 272; |
| 220 293 398 377 272 294 399 378 273; |
| 221 295 400 379 274 296 401 380 275; |
| 222 296 401 380 275 297 402 381 276; |
| 223 297 402 381 276 298 403 382 277; |
| 224 298 403 382 277 299 404 383 278; |
| 225 299 404 383 278 300 405 384 279; |
| 226 300 405 384 279 301 406 385 280; |
| 227 301 406 385 280 302 407 386 281; |
| 228 302 407 386 281 303 408 387 282; |
| 229 303 408 387 282 304 409 388 283; |
| 230 304 409 388 283 305 410 389 284; |
| 231 305 410 389 284 306 411 390 285; |
| 232 306 411 390 285 307 412 391 286; |
| 233 307 412 391 286 308 413 392 287; |
| 234 308 413 392 287 309 414 393 288; |
| 235 309 414 393 288 310 415 394 289; |
| 236 310 415 394 289 311 416 395 290; |
| 237 311 416 395 290 312 417 396 291; |
| 238 312 417 396 291 313 418 397 292; |
| 239 313 418 397 292 314 419 398 293; |
| 240 314 419 398 293 315 420 399 294; |
| 241 337 442 421 316 338 443 422 317; |
| 242 338 443 422 317 339 444 423 318; |
Verification Examples
V.05 Solids

243 339 444 423 318 340 445 424 319;
244 340 445 424 319 341 446 425 320;
245 341 446 425 320 342 447 426 321;
246 342 447 426 321 343 448 427 322;
247 343 448 427 322 344 449 428 323;
248 344 449 428 323 345 450 429 324;
249 345 450 429 324 346 451 430 325;
250 346 451 430 325 347 452 431 326;
251 347 452 431 326 348 453 432 327;
252 348 453 432 327 349 454 433 328;
253 349 454 433 328 350 455 434 329;
254 350 455 434 329 351 456 435 330;
255 351 456 435 330 352 457 436 331;
256 352 457 436 331 353 458 437 332;
257 353 458 437 332 354 459 438 333;
258 354 459 438 333 355 460 439 334;
259 355 460 439 334 356 461 440 335;
260 356 461 440 335 357 462 441 336;
261 357 462 441 336 358 463 442 337;
262 358 463 442 337 359 464 443 338;
263 359 464 443 338 360 465 444 339;
264 360 465 444 339 361 466 445 340;
265 361 466 445 340 362 467 446 341;
266 362 467 446 341 363 468 447 342;
267 363 468 447 342 364 469 448 343;
268 364 469 448 343 365 470 449 344;
269 365 470 449 344 366 471 450 345;
270 366 471 450 345 367 472 451 346;
271 367 472 451 346 368 473 452 347;
272 368 473 452 347 369 474 453 348;
273 369 474 453 348 370 475 454 349;
274 370 475 454 349 371 476 455 350;
275 371 476 455 350 372 477 456 351;
276 372 477 456 351 373 478 457 352;
277 373 478 457 352 374 479 458 353;
278 374 479 458 353 375 480 459 354;
279 375 480 459 354 376 481 460 355;
280 376 481 460 355 377 482 461 356;
281 377 482 461 356 378 483 462 357;
282 378 483 462 357 379 484 463 358;
283 379 484 463 358 380 485 464 359;
284 380 485 464 359 381 486 465 360;
285 381 486 465 360 382 487 466 361;
286 382 487 466 361 383 488 467 362;
287 383 488 467 362 384 489 468 363;
288 384 489 468 363 385 490 469 364;
289 385 490 469 364 386 491 470 365;
290 386 491 470 365 387 492 471 366;
291 387 492 471 366 388 493 472 367;
292 388 493 472 367 389 494 473 368;
293 389 494 473 368 390 495 474 369;
294 390 495 474 369 391 496 475 370;
295 391 496 475 370 392 497 476 371;
296 392 497 476 371 393 498 477 372;
297 393 498 477 372 394 499 478 373;
298 394 499 478 373 395 500 479 374;
299 395 500 479 374 396 501 480 375;
300 396 501 480 375 397 502 481 376;
| 301 400 505 484 379 401 506 485 380; |
| 302 401 506 485 380 402 507 486 381; |
| 303 402 507 486 381 403 508 487 382; |
| 304 403 508 487 382 404 509 488 383; |
| 305 404 509 488 383 405 510 489 384; |
| 306 405 510 489 384 406 511 490 385; |
| 307 406 511 490 385 407 512 491 386; |
| 308 407 512 491 386 408 513 492 387; |
| 309 408 513 492 387 409 514 493 388; |
| 310 409 514 493 388 410 515 494 389; |
| 311 410 515 494 389 411 516 495 390; |
| 312 411 516 495 390 412 517 496 391; |
| 313 412 517 496 391 413 518 497 392; |
| 314 413 518 497 392 414 519 498 393; |
| 315 414 519 498 393 415 520 499 394; |
| 316 415 520 499 394 416 521 500 395; |
| 317 416 521 500 395 417 522 501 396; |
| 318 417 522 501 396 418 523 502 397; |
| 319 418 523 502 397 419 524 503 398; |
| 320 419 524 503 398 420 525 504 399; |

MEMBER PROPERTY AMERICAN
1001 TO 1004 PRIS YD 0.2
DEFINE MATERIAL START
  ISOTROPIC MATERIAL1
  E 1e+07
  POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
  1 106 211 316 421 PINNED
  22 127 232 337 442 PINNED
  43 148 253 358 463 PINNED
  64 169 274 379 484 PINNED
  85 190 295 400 505 PINNED
LOAD 1
  JOINT LOAD
  21 FX -48
  42 FX -48
  63 FX -48
  84 FX -48
  105 FX -48
  126 FX -48
  147 FX -48
  168 FX -48
  189 FX -48
  210 FX -48
  231 FX -48
  252 FX -48
  273 FX -48
  294 FX -48
  315 FX -48
  336 FX -48
  357 FX -48
  378 FX -48
  399 FX -48
  420 FX -48
  441 FX -48
LOAD 2
JOINT LOAD
231 FY 1000
315 FY -1000
LOAD 3
JOINT LOAD
21 FX 133.33
42 FX 133.33
63 FX 133.33
84 FX 133.33
105 FX 133.33
126 FX 66.67
147 FX 66.67
168 FX 66.67
189 FX 66.67
210 FX 66.67
336 FX -66.67
357 FX -66.67
378 FX -66.67
399 FX -66.67
420 FX -66.67
441 FX -133.33
462 FX -133.33
483 FX -133.33
504 FX -133.33
525 FX -133.33
LOAD 4
JOINT LOAD
21 FY 40
42 FY 40
63 FY 40
84 FY 40
105 FY 40
126 FY 40
147 FY 40
168 FY 40
189 FY 40
210 FY 40
231 FY 40
252 FY 40
273 FY 40
294 FY 40
315 FY 40
336 FY 40
357 FY 40
378 FY 40
399 FY 40
420 FY 40
441 FY 40
462 FY 40
483 FY 40
504 FY 40
525 FY 40
PERFORM ANALYSIS
LOAD LIST 1
PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 - 231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
LOAD LIST 2
PRINT JOINT DISPLACEMENTS LIST 21 105 273 441 525
LOAD LIST 3
PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 - 231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
LOAD LIST 4
PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 - 231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
LOAD LIST ALL
PRINT ELEMENT JOINT STRESSES SOLID LIST 10
*PRINT ELEMENT JOINT STRESSES SOLID LIST 10 TO 70 BY 20
*PRINT ELEMENT JOINT STRESSES SOLID LIST 90 TO 150 BY 20
*PRINT ELEMENT JOINT STRESSES SOLID LIST 170 TO 230 BY 20
*PRINT ELEMENT JOINT STRESSES SOLID LIST 250 TO 310 BY 20
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>1</td>
<td>-0.00033</td>
<td>-0.00002</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00002</td>
<td>-0.00003</td>
</tr>
<tr>
<td>42</td>
<td>1</td>
<td>-0.00031</td>
<td>-0.00002</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>63</td>
<td>1</td>
<td>-0.00031</td>
<td>-0.00002</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>84</td>
<td>1</td>
<td>-0.00031</td>
<td>-0.00002</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>105</td>
<td>1</td>
<td>-0.00033</td>
<td>-0.00002</td>
<td>0.00001</td>
<td>-0.00000</td>
<td>-0.00002</td>
<td>-0.00003</td>
</tr>
<tr>
<td>126</td>
<td>1</td>
<td>-0.00030</td>
<td>-0.00001</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>147</td>
<td>1</td>
<td>-0.00029</td>
<td>-0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>168</td>
<td>1</td>
<td>-0.00029</td>
<td>-0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>189</td>
<td>1</td>
<td>-0.00029</td>
<td>-0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>210</td>
<td>1</td>
<td>-0.00030</td>
<td>-0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>231</td>
<td>1</td>
<td>-0.00030</td>
<td>0.00000</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>252</td>
<td>1</td>
<td>-0.00029</td>
<td>0.00000</td>
<td>-0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>273</td>
<td>1</td>
<td>-0.00029</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>294</td>
<td>1</td>
<td>-0.00029</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>315</td>
<td>1</td>
<td>-0.00030</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>336</td>
<td>1</td>
<td>-0.00030</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>357</td>
<td>1</td>
<td>-0.00029</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>378</td>
<td>1</td>
<td>-0.00029</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>399</td>
<td>1</td>
<td>-0.00029</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>420</td>
<td>1</td>
<td>-0.00030</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>441</td>
<td>1</td>
<td>-0.00033</td>
<td>0.00000</td>
<td>-0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>462</td>
<td>1</td>
<td>-0.00031</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>483</td>
<td>1</td>
<td>-0.00031</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>504</td>
<td>1</td>
<td>-0.00031</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>525</td>
<td>1</td>
<td>-0.00033</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>

************* END OF LATEST ANALYSIS RESULT *************
### Verification Examples

**V.05 Solids**

---

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAn</th>
<th>Y-RotAn</th>
<th>Z-RotAn</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>2</td>
<td>-0.0002</td>
<td>0.00149</td>
<td>-0.00224</td>
<td>0.00149</td>
<td>0.00001</td>
<td>-0.00001</td>
</tr>
<tr>
<td>105</td>
<td>2</td>
<td>0.00002</td>
<td>-0.00149</td>
<td>-0.00224</td>
<td>0.00149</td>
<td>0.00001</td>
<td>0.00001</td>
</tr>
<tr>
<td>273</td>
<td>2</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00149</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>441</td>
<td>2</td>
<td>0.00002</td>
<td>0.00149</td>
<td>0.00224</td>
<td>0.00149</td>
<td>-0.00001</td>
<td>-0.00001</td>
</tr>
<tr>
<td>525</td>
<td>2</td>
<td>-0.00002</td>
<td>0.00149</td>
<td>0.00224</td>
<td>0.00149</td>
<td>-0.00001</td>
<td>0.00001</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

509. Load List 3

510. Print Joint Displacements List 21 42 63 84 105 126 147 168 189 210

511. 231 252 273 294 315 336 357 378 399 420 441 462 483 504 525

---

A CANTILEVER BEAM WITH SOLID ELEMENTS

---

### Joint Displacement (Inch Radians)  Structure Type = SPACE

---

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAn</th>
<th>Y-RotAn</th>
<th>Z-RotAn</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>3</td>
<td>0.00132</td>
<td>0.00621</td>
<td>0.00004</td>
<td>-0.00002</td>
<td>0.00000</td>
<td>0.00248</td>
</tr>
<tr>
<td>42</td>
<td>3</td>
<td>0.00125</td>
<td>0.00621</td>
<td>0.00002</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>63</td>
<td>3</td>
<td>0.00125</td>
<td>0.00621</td>
<td>-0.00002</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>84</td>
<td>3</td>
<td>0.00132</td>
<td>0.00621</td>
<td>-0.00004</td>
<td>0.00002</td>
<td>0.00000</td>
<td>0.00088</td>
</tr>
<tr>
<td>105</td>
<td>3</td>
<td>0.00061</td>
<td>0.00619</td>
<td>0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>126</td>
<td>3</td>
<td>0.00060</td>
<td>0.00619</td>
<td>0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>147</td>
<td>3</td>
<td>0.00060</td>
<td>0.00619</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>168</td>
<td>3</td>
<td>0.00060</td>
<td>0.00619</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>189</td>
<td>3</td>
<td>0.00060</td>
<td>0.00619</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>210</td>
<td>3</td>
<td>0.00061</td>
<td>0.00619</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>231</td>
<td>3</td>
<td>0.00000</td>
<td>0.00618</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>252</td>
<td>3</td>
<td>0.00000</td>
<td>0.00618</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>273</td>
<td>3</td>
<td>0.00000</td>
<td>0.00619</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>294</td>
<td>3</td>
<td>0.00000</td>
<td>0.00618</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>315</td>
<td>3</td>
<td>0.00000</td>
<td>0.00618</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>336</td>
<td>3</td>
<td>-0.00061</td>
<td>0.00619</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>357</td>
<td>3</td>
<td>-0.00060</td>
<td>0.00619</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>378</td>
<td>3</td>
<td>-0.00060</td>
<td>0.00619</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>399</td>
<td>3</td>
<td>-0.00060</td>
<td>0.00619</td>
<td>0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>420</td>
<td>3</td>
<td>-0.00061</td>
<td>0.00619</td>
<td>0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>441</td>
<td>3</td>
<td>-0.00132</td>
<td>0.00621</td>
<td>-0.00004</td>
<td>-0.00002</td>
<td>0.00000</td>
<td>0.00088</td>
</tr>
<tr>
<td>462</td>
<td>3</td>
<td>-0.00125</td>
<td>0.00621</td>
<td>-0.00002</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>483</td>
<td>3</td>
<td>-0.00125</td>
<td>0.00621</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>504</td>
<td>3</td>
<td>-0.00125</td>
<td>0.00621</td>
<td>0.00002</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>525</td>
<td>3</td>
<td>-0.00132</td>
<td>0.00621</td>
<td>0.00004</td>
<td>0.00002</td>
<td>0.00000</td>
<td>0.00088</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

512. Load List 4

513. Print Joint Displacements List 21 42 63 84 105 126 147 168 189 210

514. 231 252 273 294 315 336 357 378 399 420 441 462 483 504 525

---

A CANTILEVER BEAM WITH SOLID ELEMENTS

---

### Joint Displacement (Inch Radians)  Structure Type = SPACE

---

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAn</th>
<th>Y-RotAn</th>
<th>Z-RotAn</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>4</td>
<td>0.00372</td>
<td>0.02537</td>
<td>-0.00001</td>
<td>0.00003</td>
<td>0.00000</td>
<td>0.00248</td>
</tr>
<tr>
<td>42</td>
<td>4</td>
<td>0.00373</td>
<td>0.02536</td>
<td>-0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>63</td>
<td>4</td>
<td>0.00373</td>
<td>0.02535</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>84</td>
<td>4</td>
<td>0.00373</td>
<td>0.02536</td>
<td>0.00001</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>105</td>
<td>4</td>
<td>0.00372</td>
<td>0.02537</td>
<td>0.00001</td>
<td>-0.00003</td>
<td>0.00000</td>
<td>0.00248</td>
</tr>
<tr>
<td>126</td>
<td>4</td>
<td>0.00184</td>
<td>0.02535</td>
<td>0.00001</td>
<td>-0.00003</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>
515. LOAD LIST ALL
516. PRINT ELEMENT JOINT STRESSES SOLID LIST 10
ELEMENT JOINT STRESSES SOLID
A CANTILEVER BEAM WITH SOLID ELEMENTS

ELEMENT STRESSES UNITS= POUNINCH

<table>
<thead>
<tr>
<th>NODE/LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SXX</td>
<td>SYY</td>
</tr>
<tr>
<td></td>
<td>SXZ</td>
<td>SYY</td>
</tr>
</tbody>
</table>

10  1  31  -200.005  -0.003  0.000  0.001  0.000
   -0.001
10  1  136  -199.999  -0.003  0.001  0.001  -0.000
   -0.000
10  1  115  -200.000  -0.003  0.002  0.001  -0.000
   -0.000
10  1  10  -200.007  -0.003  -0.000  0.001  -0.000
   -0.001
10  1  32  -200.004  -0.001  0.001  0.001  -0.000
   -0.000
10  1  137  -199.999  -0.001  0.000  0.001  0.000
   -0.001
10  1  116  -200.000  -0.002  0.000  0.001  0.000
   -0.000
10  1  11  -200.007  -0.002  0.001  0.001  0.000
   -0.001
10  1 CENTER  -200.003  -0.002  0.001  0.001  -0.000
   -0.000

S1=  0.001  S2=  -0.002  S3=  -200.003  SE= 200.002
DC=  -0.000  -0.017  1.000  0.000  1.000
0.017
10  2  31  0.006  -0.006  -0.008  137.619  0.001
### Verification Examples

**V.05 Solids**

---

<table>
<thead>
<tr>
<th>NODE</th>
<th>ELEMENT LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>3</td>
<td>833.325</td>
<td>-0.002</td>
<td>-0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>416.662</td>
<td>-0.002</td>
<td>-0.000</td>
<td>-0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>416.662</td>
<td>-0.002</td>
<td>-0.001</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>833.325</td>
<td>-0.002</td>
<td>-0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>833.325</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>416.662</td>
<td>0.001</td>
<td>0.000</td>
<td>-0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>416.662</td>
<td>0.001</td>
<td>0.000</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>833.325</td>
<td>0.001</td>
<td>0.000</td>
<td>-0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>10</td>
<td>3 CENTER</td>
<td>624.994</td>
<td>-0.001</td>
<td>-0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

**S1= 624.994  S2= 0.000  S3= -312.382  SE= 624.994**

**DC= 1.000  0.000  0.000  0.000  0.000  1.000**

---

**A CANTILEVER BEAM WITH SOLID ELEMENTS**

---

**STAAD.Pro User Manual**
V.06 Loading

V. IBC / ASCE 7

V. IBC 2012

V. IBC 2012 Response Spectrum

Reference
1. Example 4.7, http://nptel.ac.in/courses/105101004/4 (p.36)

Related Links
• TR.32.10.1.9 Response Spectrum Specification per IBC 2012 (on page 2748)

Details
Example 4.7(1) is modelled in STAAD.Pro considering the following:

Location: Corona, CA (ZIP 92887)
Short period acceleration, $S_s = 2.24658$
One second acceleration: $S_t = 0.817039$
Site class E
Response modification factor, $R = 3$
Importance factor, $I = 1.0$
The generated model is subjected to a Response Spectrum Load along global X direction. The base shear reported by STAAD.Pro is verified against hand calculation.

**Validation**

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

**Table 425: Natural Frequencies/Time Period**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (cyc/sec)</th>
<th>Time Period (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

**Table 426: Mode Shapes**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Story Level/ Joint Numbers</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>-0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
Table 427: Mode Participation Factor Calculation

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$\phi$</td>
<td>$Wi \times \phi$</td>
<td>$Wi \times \phi^2$</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>49.035</td>
<td>49.035</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86603</td>
<td>84.932</td>
<td>73.553</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>49.035</td>
<td>24.518</td>
</tr>
<tr>
<td>Summation</td>
<td>245.175</td>
<td>183.00</td>
<td>147.11</td>
<td>147.105</td>
</tr>
</tbody>
</table>

Modal Weight Mk × g
- Mode 1: 227.66
- Mode 2: 16.35
- Mode 3: 1.173

Modal Weight Participation (In %)
- Mode 1: 92.85
- Mode 2: 6.667
- Mode 3: 0.479

Mode Participation Factor $\prod \frac{\sum Wi \times \phi}{\sum Wi \times \phi^2}$
- Mode 1: 1.244
- Mode 2: 0.333
- Mode 3: 0.0893

Horizontal Acceleration Spectrum Value Calculation.

$S_{ms} = S_a \times F_a = 2.24658 \times 0.9 = 2.022$

$S_{m1} = S_1 \times F_v = 0.817039 \times 2.4 = 1.961$

$S_{ds} = 2/3 \times S_{ms} = 2/3 \times 2.022 = 1.348$

$S_{d1} = 2/3 \times S_{m1} = 2/3 \times 1.961 = 1.307$

$T_0 = 0.2 \times (S_{d1}/S_{ds}) = 0.2 \times (1.307 / 1.348) = 0.1940$

$T_s = S_{d1}/S_{ds} = 1.307 / 1.348 = 0.9698$

<table>
<thead>
<tr>
<th>Mode</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time Period T</td>
<td>0.30014 ($T_0 \leq T \leq T_s$)</td>
<td>0.10986 ($T &lt; T_0$)</td>
<td>0.08042 ($T &lt; T_0$)</td>
</tr>
</tbody>
</table>
**Verification Examples**

V.06 Loading

<table>
<thead>
<tr>
<th><strong>Story Level</strong></th>
<th><strong>Weight Wi (KN)</strong></th>
<th><strong>Mode 1</strong></th>
<th><strong>Mode 2</strong></th>
<th><strong>Mode 3</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$S_a = S_{ds}$</td>
<td>$S_a = S_{ds} \times (0.4 + 0.6 \times \left(\frac{T}{T_0}\right))$</td>
<td>$S_a = S_{ds} \times (0.4 + 0.6 \times \left(\frac{T}{T_0}\right))$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$S_a$</td>
<td>1.348</td>
<td>0.997</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$C_s = S_a/(R_s/I)$</td>
<td>0.449</td>
<td>0.332</td>
</tr>
</tbody>
</table>

Table 428: Base Shear Calculation and its Distribution Across Height

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>$\phi$</th>
<th>$Qi (C_s \times \phi \times \sum Wi)$</th>
<th>$Vi (\sum Qi)$</th>
<th>$\phi$</th>
<th>$Qi (C_s \times \phi \times \sum Wi)$</th>
<th>$Vi (\sum Qi)$</th>
<th>Story Shear due to all modes i.e. SRSS of all modes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>27.408</td>
<td>27.408</td>
<td>-1</td>
<td>-5.433</td>
<td>-5.433</td>
<td>1</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.8</td>
<td>66 3</td>
<td>47.473</td>
<td>74.881</td>
<td>0</td>
<td>0</td>
<td>-5.433</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>27.408</td>
<td>102.29</td>
<td>1</td>
<td>10.867</td>
<td>5.433</td>
<td>0.5</td>
</tr>
<tr>
<td>Base Shear, $V$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Comparison**

Table 429: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>STAAD.Pro</th>
<th>Reference</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base shear (kN)</td>
<td>102.43</td>
<td>102.33</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\2012\IBC 2012 Response Spectrum.STD is typically installed with the program.

`STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR`
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS IBC 2012 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>MODE</th>
<th>CALCULATED FREQUENCIES FOR LOAD CASE 1</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FREQUENCY(CYCLES/SEC)</td>
</tr>
</tbody>
</table>

*Verification Examples*

*V.06 Loading*

**STAAD.Pro**
<table>
<thead>
<tr>
<th>Mode</th>
<th>Period</th>
<th>FX (KN)</th>
<th>FY (KN)</th>
<th>FZ (KN)</th>
<th>MX (KN)</th>
<th>MY (KN)</th>
<th>MZ (KN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.300</td>
<td>102.19</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-613.12</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.110</td>
<td>5.43</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>16.28</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.080</td>
<td>0.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2.05</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**V. IBC 2012 Static Seismic**

Calculation of base shear and its distribution along the height for equivalent lateral force method in IBC 2012.
**Problem**

Structure to be modelled:

- height = 4 × 4 m = 16 m
- plan dimension X = 4 m
- plan dimension Z = 5 m

Input parameters:

- Location: Corona, CA, USA (ZIP 92877)
- Site class D (SCLASS 4)
- Long-period transition time = 12 s (TL 12)
- Importance factor = 1.0 (I 1.0)
- Response modification factor X = 3 (RX 3)
- Response modification factor Z = 4 (RZ 4)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50 kN to nodes 2, 3, 5, 6, 8, 9, 11 To 20. Seismic Load as per IBC 2012 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

**Calculations**

ZIP = 92887; $S_s = 2.247$ & $S_1 = 0.817$

For Site Class D, $F_a = 1$ ($S_s > 1.25$) and $F_v = 1.5$ ($S_1 > 0.5$)

$SMS = S_s \times F_a = 2.247$, $SM1 = S_1 \times F_v = 1.2255$

$SDS = 2/3 \times SMS = 1.498$, $SD1 = 2/3 \times SM1 = 0.817$

For concrete moment resisting frames $Ct = 0.0466$ in metric system & $x = 0.9$

Height = $ha = 16 m$, $Ta = Ct \times ha^x = 0.0466 \times 16^{0.9} = 0.565$,

Since $SD1 = 0.817 > 0.4$, $Cu = 1.4$

$Cu \times Ta = 1.4 \times 0.565 = 0.791$

$T$ (from output file) = 1.286, $Cu \times Ta = 0.791 < 1.286$, $T_{used} = 0.791$

$Cs_x = SDS/(Rx/I) = 1.498 / (3/1) = 0.49933$

$Cs_x$ lower limit = $0.044 \times SDS \times I & 0.5 \times S1 / (Rx/I)$.

$Cs_x$ lower limit = $0.044 \times 1.498 \times 1 & 0.5 \times 0.817 / (3/1)$

$Cs_x$ lower limit = $0.065912 & 0.136166$

$Cs_x$ upper limit = $SD1 / (T_{used} \times (Rx/I))$ as $T_{used} < TL$

$Cs_x$ upper limit = $0.817 / (0.791 \times (3/1)) = 0.34429$

$Cs_x$ used = 0.34429

Seismic Base Shear $V = Cs \times W = 0.34429 \times (200 \times 4) = 275.432$ kN

Lateral seismic force $Fx = C_{vx} \times V$ where $C_{vx} = W_x \times h^x / \sum W_i \times h^i$
Since $T_{used} = 0.791 \text{ sec} > 0.5 \text{ sec}$ distribution of Base Shear along story levels would have the value of exponent $k$ linearly interpolated between 1 (less than equal to 0.5 sec) and 2 (greater than equal to 2.5 sec) i.e. power = 1.1455.

<table>
<thead>
<tr>
<th>Story Level</th>
<th>$W_i$</th>
<th>$h_i$</th>
<th>$Wi \times h_i^{1.1455}$</th>
<th>$(Wi \times h_i^{1.1455})/\sum(Wi \times h_i^{1.1455})$</th>
<th>$F_x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>200</td>
<td>16</td>
<td>4,790.2</td>
<td>0.4209</td>
<td>115.941</td>
</tr>
<tr>
<td>3rd</td>
<td>200</td>
<td>12</td>
<td>3,445.3</td>
<td>0.3028</td>
<td>83.391</td>
</tr>
<tr>
<td>2nd</td>
<td>200</td>
<td>8</td>
<td>2,165.3</td>
<td>0.1903</td>
<td>52.409</td>
</tr>
<tr>
<td>1st</td>
<td>200</td>
<td>4</td>
<td>978.79</td>
<td>0.0860</td>
<td>23.691</td>
</tr>
<tr>
<td>Summation</td>
<td>800</td>
<td></td>
<td>11,379.6</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>$V_x$</td>
<td></td>
<td></td>
<td>275.432</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Comparison**

**Table 430: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear, $F_x$ (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1st</td>
<td>23.691</td>
<td>23.697</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>2nd</td>
<td>52.409</td>
<td>52.389</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>3rd</td>
<td>83.391</td>
<td>83.364</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>roof</td>
<td>115.941</td>
<td>115.908</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Base shear, $V_x$ (kN)</td>
<td>275.432</td>
<td>275.34</td>
<td>Negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\2012\IBC 2012 Static Seismic.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL

MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25

CONSTANTS
MATERIAL CONCRETE ALL

SUPPORTS
1 4 7 10 FIXED

*DEFINE IBC 2015
DEF IBC 2012
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 4 TL 12
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50

*SELFWEIGHT 1
*MEMBER WEIGHT
2 5 7 TO 10 12 15 UNI 10
LOAD 1 LOADTYPE Seismic TITLE EL-X DIR
IBC LOAD X 1

PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH

STAAD Output

*********************************************************
* IBC 2012 SEISMIC LOAD ALONG X :                      *
* CT =  0.047 Cu =  1.400 x =  0.9000                  *
* TIME PERIODS :                                        *
* Ta =  0.565 T =  1.286 Tuser =  0.000                *
* TIME PERIOD USED (T) =  0.791                         *
* Cs LIMITS : LOWER =  0.136 UPPER =  0.344            *
* LOAD FACTOR =  1.000                                 *
* DESIGN BASE SHEAR =  1.000 X  0.344 X       800.00   *
* =       275.34  KN                                  *
*********************************************************

JOINT        LATERAL LOAD (KN )  TORSIONAL MOMENT (KN -METE) FACTOR - 1
--------------- ------- ------- ------- ------- ------ -------
    2  FX  5.920  MY  0.000
    3  FX  5.920  MY  0.000
    8  FX  5.920  MY  0.000
    9  FX  5.920  MY  0.000
--------------- ------- ------- ------- ------- ------ -------
### Related Links
- [TR.31.2.13 IBC 2012 Seismic Load Definition](on page 2596)

### V. IBC 2015

#### V. IBC 2015 Response Spectrum


**Reference**

1. Example 4.7, [http://nptel.ac.in/courses/105101004/4](http://nptel.ac.in/courses/105101004/4) (p.36)

**Related Links**
- [TR.32.10.1.10 Response Spectrum Specification per IBC 2015](on page 2754)

#### Details

Example 4.7(1) is modelled in STAAD.Pro considering the following:

- **Location:** Corona, CA (ZIP 92887)
- **Short period acceleration,** $S_s = 2.24658$
- **One second acceleration:** $S_1 = 0.817039$
- **Site class E**
- **Response modification factor,** $R = 3$
- **Importance factor,** $I = 1.0$
- $F_a = 0.9$
- $F_v = 2.4$
- **TL = 8**

The generated model is subjected to a Response Spectrum Load along global X direction. The base shear reported by STAAD.Pro is verified against hand calculation.
**Validation**

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

**Table 431: Natural Frequencies/Time Period**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (cyc/sec)</th>
<th>Time Period (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

**Table 432: Mode Shapes**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Story Level/ Joint Numbers</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>-0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
### Table 433: Mode Participation Factor Calculation

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight $Wi$ (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\phi$</td>
<td>$Wi \times \phi$</td>
<td>$Wi \times \phi^2$</td>
<td>$\phi$</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>49.035</td>
<td>49.035</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86603</td>
<td>84.932</td>
<td>73.553</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>49.035</td>
<td>24.518</td>
</tr>
<tr>
<td>Summation</td>
<td>245.175</td>
<td>183.00</td>
<td>147.11</td>
<td>49.035</td>
</tr>
<tr>
<td>Modal Weight $Mk \times g$</td>
<td>227.66</td>
<td>16.35</td>
<td>1.173</td>
<td></td>
</tr>
<tr>
<td>Modal Weight Participation (In %)</td>
<td>92.85</td>
<td>6.667</td>
<td>0.479</td>
<td></td>
</tr>
<tr>
<td>Mode Participation Factor $\prod \sum Wi \times \phi / \sum Wi \times \phi^2$</td>
<td>1.244</td>
<td>0.333</td>
<td>0.0893</td>
<td></td>
</tr>
</tbody>
</table>

#### Horizontal Acceleration Spectrum Value Calculation.

- $S_{ms} = S_a \times F_s = 2.24658 \times 0.9 = 2.022$
- $S_{m1} = S_1 \times F_v = 0.817039 \times 2.4 = 1.961$
- $S_{ds} = 2/3 \times S_{ms} = 2/3 \times 2.022 = 1.348$
- $S_{d1} = 2/3 \times S_{m1} = 2/3 \times 1.961 = 1.307$
- $T_0 = 0.2 \times (S_{d1}/S_{ds}) = 0.2 \times (1.307 / 1.348) = 0.1940$
- $T_s = S_{d1}/S_{ds} = 1.307 / 1.348 = 0.9698$

<table>
<thead>
<tr>
<th></th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time Period $T$</td>
<td>0.30014 ($T_0 \leq T \leq T_s$)</td>
<td>0.10986 ($T &lt; T_0$)</td>
<td>0.08042 ($T &lt; T_0$)</td>
</tr>
<tr>
<td>Formula</td>
<td>$S_a = S_{ds}$</td>
<td>$S_a = S_{ds} \times (0.4 + 0.6 \times (T/T_0))$</td>
<td>$S_a = S_{ds} \times (0.4 + 0.6 \times (T/T_0))$</td>
</tr>
<tr>
<td>$S_a$</td>
<td>1.348</td>
<td>0.997</td>
<td>0.875</td>
</tr>
<tr>
<td>$C_s = S_a/(R_s/I)$</td>
<td>0.449</td>
<td>0.332</td>
<td>0.292</td>
</tr>
</tbody>
</table>
Table 434: Base Shear Calculation and its Distribution Across Height

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Story Shear due to all modes i.e. SRSS of all modes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>φ</td>
<td>Qi(Cs × φ × ∏ × Wi)</td>
<td>Vi (∑Qi)</td>
<td>Qi(Cs × φ × ∏ × Wi)</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>27.408</td>
<td>27.408</td>
<td>-1</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86</td>
<td>47.473</td>
<td>74.881</td>
<td>0</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>27.408</td>
<td>102.29</td>
<td>1</td>
</tr>
</tbody>
</table>

Base Shear, V 102.43

Comparison

Table 435: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>STAAD.Pro</th>
<th>Reference</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base shear (kN)</td>
<td>102.43</td>
<td>102.33</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Pro User Manual

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\V.06 Loading\IBC\2015\IBC 2015 Response Spectrum.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0; 2 0 3; 3 3 0; 4 3 0; 5 0 6; 6 3 6; 7 0 9; 8 3 9; 1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
MEMBER INCIDENCES
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
Verification Examples

V.06 Loading

TABLE 2
UNIT METER KN
PRISMATIC BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS IBC 2015 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

RESPONSE SPECTRUM LOAD 1
RESPONSE LOAD CASE 1
MODAL WEIGHT (MODAL MASS TIMES g) IN KN GENERALIZED

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.27656E+02</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>2</td>
<td>1.63450E+01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>3</td>
<td>1.173519E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471051E+02</td>
</tr>
</tbody>
</table>

SRSS MODAL COMBINATION METHOD USED.
### Dynamic Weight X Y Z
- X: 2.451750E+02
- Y: 0.000000E+00
- Z: 0.000000E+00 KN

### Missing Weight X Y Z
- X: -3.177132E-06
- Y: 0.000000E+00
- Z: 0.000000E+00 KN

### Modal Weight X Y Z
- X: 2.451750E+02
- Y: 0.000000E+00
- Z: 0.000000E+00 KN

### Modal Base Actions

<table>
<thead>
<tr>
<th>Mode</th>
<th>Acceleration-G</th>
<th>Damping</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.34795</td>
<td>0.05000</td>
</tr>
<tr>
<td>2</td>
<td>0.99725</td>
<td>0.05000</td>
</tr>
<tr>
<td>3</td>
<td>0.87451</td>
<td>0.05000</td>
</tr>
</tbody>
</table>

### Modal Base Actions

<table>
<thead>
<tr>
<th>Mode</th>
<th>Forces in KN</th>
<th>Length in Mete</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Momentum are</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mode</th>
<th>Period</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
<th>MY</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.300</td>
<td>102.19</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-613.12</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.110</td>
<td>5.43</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>16.28</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.080</td>
<td>0.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2.05</td>
<td></td>
</tr>
</tbody>
</table>

### Participation Factors

<table>
<thead>
<tr>
<th>Mode</th>
<th>Mass</th>
<th>Participation Factors in Percent</th>
<th>Base Shear in KN</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
<td>Y</td>
<td>Z</td>
</tr>
<tr>
<td>1</td>
<td>92.85</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>6.67</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.48</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mode</th>
<th>Total SRSS Shear</th>
<th>102.33</th>
<th>0.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>Total 10PCT Shear</td>
<td>102.33</td>
<td>0.00</td>
</tr>
<tr>
<td>0.00</td>
<td>Total ABS Shear</td>
<td>107.96</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### V. IBC 2015 Static Seismic

Calculation of base shear and its distribution along the height for equivalent lateral force method in IBC 2015.

**Problem**

Structure to be modelled:

- height = 4 x 4 m = 16 m
- plan dimension X = 4 m
- plan dimension Z = 5 m

Input parameters:
Verification Examples
V.06 Loading

Location: Corona, CA, USA (ZIP 92877)
Site class D (SCLASS 4)
Long-period transition time = 12s (TL 12)
Importance factor = 1.0 (I 1.0)
Response modification factor X = 3 (RX 3)
Response modification factor Z = 4 (RZ 4)

The seismic weight for each floor is assumed as 200 KN applied as joint weight of 50KN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per IBC 2015 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations
ZIP = 92887; Ss = 2.247 & S1 = 0.817
For Site Class D, Fa = 1 (Ss > 1.25) and Fv = 1.5 (S1 > 0.5)
SMS = Ss × Fa = 2.247, SM1 = S1 × Fv = 1.2255
SDS = 2/3 × SMS = 1.498, SD1 = 2/3 × SM1 = 0.817
For concrete moment resisting frames Ct = 0.0466 in metric system & x = 0.9
Height = ha = 16m, Ta = Ct × ha^x = 0.0466 × 16^0.9 = 0.565,
Since SD1 = 0.817 > 0.4, Cu = 1.4
Cu × Ta = 1.4 × 0.565 = 0.791
T (from output file) = 1.286, Cu × Ta = 0.791 < 1.286, Tused = 0.791
Cs_x = SDS / (Rx/I) = 1.498 / (3/1) = 0.49933
Cs_x lower limit = 0.044 × SDS × I & 0.5 × S1 / (Rx/I),
Cs_x lower limit = 0.044 × 1.498 × 1 & 0.5 × 0.817 / (3/1)
Cs_x lower limit = 0.065912 & 0.136166
Cs_x upper limit = SD1 / (Tused × (Rx/I)) as Tused < TL
Cs_x upper limit = 0.817 / (0.791 × (3/1)) = 0.34429
Cs_x used = 0.34429
Seismic Base Shear V = Cs × W = 0.34429 × (200 × 4) = 275.432 KN
Lateral seismic force Fx = Cvx × V where Cvx = Wx × hx^k / Σ Wx × hi^k
Since Tused = 0.791 sec > 0.5 Sec distribution of Base Shear along story levels would have the value of exponent k linearly interpolated between 1 (less than equal to 0.5 sec) and 2 (greater than equal to 2.5 sec) i.e. power = 1.1455.

<table>
<thead>
<tr>
<th>Story Level</th>
<th>W_i</th>
<th>h_i</th>
<th>Wi × hi 1.1455</th>
<th>(Wi × hi 1.1455) / Σ(Wi × hi 1.1455)</th>
<th>Fx</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>200</td>
<td>16</td>
<td>4,790.2</td>
<td>0.4209</td>
<td>115.941</td>
</tr>
<tr>
<td>3rd</td>
<td>200</td>
<td>12</td>
<td>3,445.3</td>
<td>0.3028</td>
<td>83.391</td>
</tr>
</tbody>
</table>

STAAD.Pro 3437 User Manual
<table>
<thead>
<tr>
<th>Story Level</th>
<th>$W_i$</th>
<th>$h_i$</th>
<th>$Wi \times hi^{1.1455}$</th>
<th>(\frac{(Wi \times hi^{1.1455})}{\sum(Wi \times hi^{1.1455})})</th>
<th>$F_x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2nd</td>
<td>200</td>
<td>8</td>
<td>2,165.3</td>
<td>0.1903</td>
<td>52.409</td>
</tr>
<tr>
<td>1st</td>
<td>200</td>
<td>4</td>
<td>978.79</td>
<td>0.0860</td>
<td>23.691</td>
</tr>
<tr>
<td>Summation</td>
<td>800</td>
<td>4</td>
<td>11,379.6</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>$V_x$</td>
<td></td>
<td>275.432</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Comparison**

Table 436: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear, $F_x$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1st</td>
<td>23.691</td>
<td>23.697</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>2nd</td>
<td>52.409</td>
<td>52.389</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>3rd</td>
<td>83.391</td>
<td>83.364</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>roof</td>
<td>115.941</td>
<td>115.908</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Base shear, $V_x$</td>
<td>275.432</td>
<td>275.34</td>
<td>Negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\2015\IBC 2015 Static Seismic.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14 18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616  
ALPHA 1e-05  
DAMP 0.05  
TYPE CONCRETE  
STRENGTH FCU 27579  
END DEFINE MATERIAL  
MEMBER PROPERTY AMERICAN  
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3  
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25  
CONSTANTS  
MATERIAL CONCRETE ALL  
SUPPORTS  
1 4 7 10 FIXED  
DEFINE IBC 2015  
*DEFINE IBC 2012  
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 4 TL 12  
JOINT WEIGHT  
2 3 5 6 8 9 11 TO 20 WEIGHT 50  
*SELFWEIGHT 1  
*MEMBER WEIGHT  
*2 5 7 TO 10 12 15 UNI 10  
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR  
IBC LOAD X 1  
PERFORM ANALYSIS PRINT LOAD DATA  
PRINT ANALYSIS RESULTS  
FINISH

**STAAD Output**

```
******************************************************************************************
* IBC 2015 SEISMIC LOAD ALONG X :                       *
* CT =  0.047 Cu = 1.400 x = 0.9000                     *
* TIME PERIODS :                                        *
* Ta = 0.565 T = 1.286 Tuser = 0.000                   *
* TIME PERIOD USED (T) = 0.791                         *
* Cs LIMITS : LOWER = 0.136 UPPER = 0.344              *
* LOAD FACTOR = 1.000                                  *
* DESIGN BASE SHEAR = 1.000 X 0.344 X 800.00 = 275.34 KN *
******************************************************************************************

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN)</th>
<th>TORSIONAL LOAD (KN)</th>
<th>LOAD - 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>FX 5.920</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>FX 5.920</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>FX 5.920</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>FX 5.920</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TOTAL = 23.679</td>
<td>0.000 AT LEVEL 4.000 MOTE</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>FX 13.097</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>FX 13.097</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>FX 13.097</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>FX 13.097</td>
<td>MY 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TOTAL = 52.389</td>
<td>0.000 AT LEVEL 8.000 MOTE</td>
<td></td>
</tr>
</tbody>
</table>
```


Verification Examples
V.06 Loading

STAAD.Pro

3439
User Manual
## Verification Examples

### V.06 Loading

<table>
<thead>
<tr>
<th></th>
<th></th>
<th>FX</th>
<th></th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
<td></td>
<td></td>
<td>20.841</td>
<td></td>
<td>0.000</td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td>20.841</td>
<td></td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 83.364</td>
<td></td>
<td>0.000 AT LEVEL</td>
<td>12.000 METE</td>
</tr>
<tr>
<td>17</td>
<td></td>
<td></td>
<td>28.977</td>
<td></td>
<td>0.000</td>
</tr>
<tr>
<td>18</td>
<td></td>
<td></td>
<td>28.977</td>
<td></td>
<td>0.000</td>
</tr>
<tr>
<td>19</td>
<td></td>
<td></td>
<td>28.977</td>
<td></td>
<td>0.000</td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
<td>28.977</td>
<td></td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 115.908</td>
<td></td>
<td>0.000 AT LEVEL</td>
<td>16.000 METE</td>
</tr>
</tbody>
</table>

### Related Links
- [TR.31.2.14 IBC 2015 Seismic Load Definition](on page 2600)

### V. IBC 2018

#### V. IBC 2018 Static Seismic T 1.2

Verify the program-calculated base shear and its distribution along the height of a three-story frame by using the equivalent lateral force method per IBC 2018. Also, verify the torsional moments to which the floors are subjected to, considering inherent as well as accidental torsion.

**Details**

A three-story structure is subject to a seismic load from the +X direction (see following figures).
Verification Examples
V.06 Loading

Figure 408: Plan View
Figure 409: Side Elevation (from +X dir)
Assumptions

i. Mapped MCER spectral response acceleration parameter at short period, $S_s = 2.02$

ii. Mapped MCER spectral response acceleration parameter at a period of 1 s, $S_1 = 0.795$

iii. Risk Category – I (Hence, From Table 1.5-2, Importance Factor $I = 1$)

iv. Site Class – D (SCLASS 4)

v. Response Modification Factor (RX & RZ) = 3

vi. Long-period Transition Time (TL) = 12 s

vii. Seismic weight is composed of UDLs (magnitude -5 kN/m, direction GY) (defined in Reference Load Definition), incident on the beams

viii. Time Period of the Structure in both directions (PX & PZ) = 1.2 s

ix. Time period coefficients, $CTX = CTZ = 0.28$

x. Exponents in time period equation: $XX = XZ = 1$

xi. The effect of shear deformation is neglected

Validation

Calculation of Base Shear

Based on $S_s$ and $S_1$, $F_o = 1$ (per table 11.4-1) and $F_v = 1.7$ (per table 11.4-2).

Hence:
\[ S_{MS} = F_s \times S_s = 1 \times 2.02 = 2.02 \]
\[ S_{DS} = 2/3 \times S_{MS} = 1.347 \]
\[ S_{M1} = F_v \times S_1 = 1.7 \times 0.795 = 1.382 \]
\[ S_{D1} = 2/3 \times S_{M1} = 0.901 \]
\[ (C_s)_{initial} = \frac{S_{DS}}{R/\gamma} = 0.4489 \]

The natural period of the structure, \( T_N = 1.2 \) s (\( T_N \) \( = \) 1.2, provides as user input).

Height of the structure, \( h = 9 \) m.

So the approximate time period, \( T_a = C_t \times h = 0.28(9m)^{1.0} = 2.52 \) s

From Table 12.8-1, \( C_u = 1.4 \) (for \( S_{D1} > 0.4 \)).

\[ C_u \times T_a = 3.528 \text{ s} \]

The time period used, \( T_R \), is the lesser of \( T_N \) and \( C_u \times T_a \), which is \( 1.2 \) s (< \( T_L = 12 \) s).

\[ (C_s)_{max} = \frac{S_{D1}}{T \times (R/\gamma)} = 0.25028 \]

\[ (C_s)_{min1} = 0.044 \times S_{DS} \times I = 0.059 \]

Since \( S_1 = 0.795 > 0.6g \), equation 12.8-6 also needs to be considered for calculating the lower limit of \( C_s \):

\[ (C_s)_{min2} = \frac{0.5 \times S_1}{R/\gamma} = 0.1325 \]

Therefore, \( (C_s)_{min} = 0.1325 < C_s \).

\[ C_s = 0.25028 \]

The seismic weight, \( W \) is taken as the total seismic weight of the beams: \( = 5 \times 3 \times 69 = 1,035 \) kN

So the total base shear, \( V = C_s \times W = 0.25 \times 1,035 = 259.04 \) kN

Vertical Distribution of Lateral Forces

Since, time period of the structure \( T = 1.2 s > 0.5 \) s and \( T < 2.5 \) s, as per clause 12.8.3, value of \( k \) needs to be linearly interpolated.

\[ k = 1.35 \]

Hence, from equation 12.8-11 and equation 12.8-12, we can find the lateral forces in different story levels, as follows:

**Table 437: Vertical force distribution**

<table>
<thead>
<tr>
<th>Story Level</th>
<th>( W_x ) (kN)</th>
<th>( h_x ) (m)</th>
<th>( W_x \times h_x )</th>
<th>( \sum_{i=1}^{3} (W_i \times h_i^k) )</th>
<th>Lateral force at story level (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>( 5 \times 3 \times 23 = 345 )</td>
<td>9</td>
<td>6,700</td>
<td>0.554</td>
<td>143.48</td>
</tr>
<tr>
<td>2nd</td>
<td>345</td>
<td>6</td>
<td>3,875</td>
<td>0.320</td>
<td>82.999</td>
</tr>
</tbody>
</table>
Consideration of Inherent Torsion (Clause 12.8.4.1)

Floor Level – Roof (9 m)
From output file, CRZ = 3.831 m and CMZ = 3.848 m
Hence, static eccentricity $e_{si} = CMZ - CRZ = 0.017$ m

Floor Level – 2nd (6 m)
From output file, CRZ = 3.858 m and CMZ = 3.848 m
Hence, static eccentricity $e_{si} = CMZ - CRZ = -0.01$ m

Floor Level – 1st (3 m)
From output file, CRZ = 3.902 m and CMZ = 3.848 m
Hence, static eccentricity $e_{si} = CMZ - CRZ = -0.054$ m

Consideration of Accidental Torsion (Clause 12.8.4.2)
At all floor levels:
\[
0.05 \times L_z = 0.05 \times 9m = 0.45 m
\]
where $L_z$ is the dimension of the structure along the global Z axis
Hence, total eccentricity to inherent and accidental torsion at roof level $e_r = (0.45 + 0.017) = 0.467$ m
Total eccentricity to inherent and accidental torsion at 2nd floor level $e_2 = (0.45 - 0.01) = 0.44$ m
Total eccentricity to inherent and accidental torsion at 1st floor level $e_1 = (0.45 - 0.054) = 0.396$ m

Total torsional moment at roof level $= F_{roof} \times e_r = 143.48 \times 0.467 = 67.005$ kN·m
Total Torsional moment at 2nd floor level $= F_{2nd} \times e_2 = 82.998 \times 0.440 = 36.519$ kN·m
Total torsional moment at 1st floor level $= F_{1st} \times e_1 = 32.559 \times 0.396 = 12.893$ kN·m

Results

Table 438: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base shear, V (kN)</td>
<td>259.04</td>
<td>259.04</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Lateral force at roof level (kN)</td>
<td>143.48</td>
<td>143.48</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
## Result Type

<table>
<thead>
<tr>
<th></th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lateral force at 2nd floor (kN)</td>
<td>82.998</td>
<td>82.998</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Lateral force at 1st floor (kN)</td>
<td>32.559</td>
<td>32.559</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Torsional moment at roof level (kN-m)</td>
<td>67.005</td>
<td>67.029</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Torsional moment at 2nd floor (kN-m)</td>
<td>36.519</td>
<td>36.493</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Torsional moment at 1st floor (kN-m)</td>
<td>12.893</td>
<td>12.875</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\2018\IBC 2018 Static Seismic T 1.2.STD is typically installed with the program.

**INPUT WIDTH 79**

**SET SHEAR**

**UNIT METER KN**

**JOINT COORDINATES**

1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 0 3; 6 0 3 3; 7 3 3 3; 8 3 0 3;
9 0 0 6; 10 0 3 6; 11 3 3 6; 12 3 0 6; 13 0 0 9; 14 0 3 9; 15 3 3 9; 16 3 0 9;
17 6 3 0; 18 6 0 0; 19 6 3 3; 20 6 0 3; 21 6 3 6; 22 6 0 6; 25 9 3 3; 26 9 0 3;
27 9 3 6; 28 9 0 6; 33 0 6 0; 35 0 6 3; 36 3 6 3; 37 0 6 6; 38 3 6 6;
39 0 6 9; 40 3 6 9; 41 6 6 0; 42 6 6 3; 43 6 6 6; 45 9 6 3; 46 9 6 6; 49 0 9 0;
50 3 9 0; 51 0 9 3; 52 3 9 3; 53 0 9 6; 54 3 9 6; 55 0 9 9; 56 3 9 9; 57 6 9 0;
58 6 9 3; 59 6 9 6; 70 9 9 3; 71 9 9 6; 72 -3 0 0; 73 -3 3 0; 74 -3 0 3;
75 -3 3 3; 76 -3 0 6; 77 -3 3 6; 80 -3 6 0; 81 -3 6 3; 82 -3 6 6; 84 -3 9 0;
85 -3 9 3; 86 -3 9 6;

**MEMBER INCIDENCES**

1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11;
11 9 10; 12 10 11; 13 11 12; 14 10 14; 15 11 15; 16 13 14; 17 14 15; 18 15 16;
19 3 17; 20 7 19; 21 11 21; 23 17 18; 24 17 19; 25 19 20; 26 19 21; 27 21 22;
30 19 25; 31 21 27; 32 25 26; 33 25 27; 34 27 28; 40 2 33; 41 3 34; 42 6 35;
43 7 36; 44 10 37; 45 11 38; 46 14 39; 47 15 40; 48 17 41; 49 19 42; 50 21 43;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 21 23 TO 27 30 TO 34 40 TO 50 52 53 56 TO 68 70 71 73 TO 75 79 TO 89 - 95 TO 107 109 110 128 129 132 TO 137 139 TO 141 143 TO 145 148 150 158 - 159 161 TO 163 165 166 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 9 12 13 16 18 20 22 26 28 72 74 76 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
MEMBER LOAD
2 4 5 7 9 10 12 14 15 17 19 TO 21 24 26 30 31 33 56 TO 68 70 71 73 TO 75 79 TO 89 - 95 TO 107 109 110 128 129 132 TO 137 139 TO 141 143 TO 145 147 TO 151 154 - 155 TO 156 158 159 161 TO 163 165 166 PRIS YD 0.4 ZD 0.4
LOAD 1 LOADTYPE Seismic  TITLE SL +X
IBC LOAD X 1 DEC 2 ACC 0.05
PERFORM ANALYSIS PRINT LOAD DATA
PRINT DIA CR
FINISH

**STAAD.Pro Output**

*****************************************************************************

---

STAAD.Pro 3447 User Manual
* EQUIV. SEISMIC LOADS AS PER IBC 2018 *
* PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD GENERATION *
* SS =  2.020 S1 =  0.795 FA =  1.000 FV = 1.700 *
* SDS =  1.347 SD1 = 0.901 *

*****************************************************************************
73. LOAD 1 LOADTYPE SEISMIC  TITLE SL +X
74. IBC LOAD X 1 DEC 2 ACC 0.05
75. PERFORM ANALYSIS PRINT LOAD DATA

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS         67  NUMBER OF MEMBERS     117
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     162
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
STAAD SPACE

4
LOADING 1 LOADTYPE SEISMIC  TITLE SL +X
*************************
* IBC 2018 SEISMIC LOAD ALONG X :
* CT =  0.280 Cu =  1.400 x =  1.000
* TIME PERIODS :
* Ta =  2.520 T =  1.200 Tuser = 1.200
* TIME PERIOD USED (T) =  1.200
* Cs LIMITS : LOWER =  0.133 UPPER =  0.250
* LOAD FACTOR          =  1.000
* DESIGN BASE SHEAR    =  1.000 X  0.250 X      1035.00   
*                       =       259.04  KN
*************************
**************************************************************************
***NOTE: SEISMIC LOAD IS ACTING AT CENTER OF MASS FOR RIGID DIAPHRAGM.
TORSION FROM STATIC ECCENTRICITY (esi) IS INCLUDED IN ANALYSIS.
DYNAMIC ECCENTRICITY APPLIED = DEC - 1
LOAD NO.:     1  DIRECTION : X   UNIT - METE

<table>
<thead>
<tr>
<th>STORY</th>
<th>LEVEL</th>
<th>DYN. ECC. (dec)</th>
<th>ACC. ECC. (aec)</th>
<th>DESIGN ECC.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.00</td>
<td>-0.05</td>
<td>-0.05</td>
<td>0.60</td>
</tr>
<tr>
<td>2</td>
<td>6.00</td>
<td>0.01</td>
<td>-0.01</td>
<td>0.60</td>
</tr>
<tr>
<td>3</td>
<td>9.00</td>
<td>0.05</td>
<td>0.02</td>
<td>0.60</td>
</tr>
</tbody>
</table>

**************************************************************************
JOINT LATERAL LOAD (KN )  TORSIONAL LOAD (KN)  LOAD - 1
----- ------- --------- --------- 1
DEC + AEC

STAAD SPACE

5
2 FX 2.123 MY 0.840
3 FX 2.123 MY 0.840
Verification Examples

V.06 Loading

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th>2.831</th>
<th>MY</th>
<th>1.120</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>FX</td>
<td>2.831</td>
<td>MY</td>
<td>1.120</td>
</tr>
<tr>
<td>7</td>
<td>FX</td>
<td>2.831</td>
<td>MY</td>
<td>1.120</td>
</tr>
<tr>
<td>10</td>
<td>FX</td>
<td>2.831</td>
<td>MY</td>
<td>1.120</td>
</tr>
<tr>
<td>11</td>
<td>FX</td>
<td>2.831</td>
<td>MY</td>
<td>1.120</td>
</tr>
<tr>
<td>14</td>
<td>FX</td>
<td>1.416</td>
<td>MY</td>
<td>0.560</td>
</tr>
<tr>
<td>15</td>
<td>FX</td>
<td>1.416</td>
<td>MY</td>
<td>0.560</td>
</tr>
<tr>
<td>17</td>
<td>FX</td>
<td>1.416</td>
<td>MY</td>
<td>0.560</td>
</tr>
<tr>
<td>19</td>
<td>FX</td>
<td>2.831</td>
<td>MY</td>
<td>1.120</td>
</tr>
<tr>
<td>21</td>
<td>FX</td>
<td>2.123</td>
<td>MY</td>
<td>0.840</td>
</tr>
<tr>
<td>25</td>
<td>FX</td>
<td>1.416</td>
<td>MY</td>
<td>0.560</td>
</tr>
<tr>
<td>27</td>
<td>FX</td>
<td>1.416</td>
<td>MY</td>
<td>0.560</td>
</tr>
<tr>
<td>73</td>
<td>FX</td>
<td>1.416</td>
<td>MY</td>
<td>0.560</td>
</tr>
<tr>
<td>75</td>
<td>FX</td>
<td>2.123</td>
<td>MY</td>
<td>0.840</td>
</tr>
<tr>
<td>77</td>
<td>FX</td>
<td>1.416</td>
<td>MY</td>
<td>0.560</td>
</tr>
<tr>
<td></td>
<td></td>
<td>-------</td>
<td>----</td>
<td>-------</td>
</tr>
<tr>
<td></td>
<td>TOTAL = 32.559</td>
<td>12.875 AT LEVEL 3.000 METE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>FX</td>
<td>5.413</td>
<td>MY</td>
<td>2.380</td>
</tr>
<tr>
<td>34</td>
<td>FX</td>
<td>5.413</td>
<td>MY</td>
<td>2.380</td>
</tr>
<tr>
<td>35</td>
<td>FX</td>
<td>7.217</td>
<td>MY</td>
<td>3.173</td>
</tr>
<tr>
<td>36</td>
<td>FX</td>
<td>7.217</td>
<td>MY</td>
<td>3.173</td>
</tr>
<tr>
<td>37</td>
<td>FX</td>
<td>7.217</td>
<td>MY</td>
<td>3.173</td>
</tr>
<tr>
<td>38</td>
<td>FX</td>
<td>7.217</td>
<td>MY</td>
<td>3.173</td>
</tr>
<tr>
<td>39</td>
<td>FX</td>
<td>3.609</td>
<td>MY</td>
<td>1.587</td>
</tr>
<tr>
<td>40</td>
<td>FX</td>
<td>3.609</td>
<td>MY</td>
<td>1.587</td>
</tr>
<tr>
<td>41</td>
<td>FX</td>
<td>3.609</td>
<td>MY</td>
<td>1.587</td>
</tr>
<tr>
<td>42</td>
<td>FX</td>
<td>7.217</td>
<td>MY</td>
<td>3.173</td>
</tr>
<tr>
<td>43</td>
<td>FX</td>
<td>5.413</td>
<td>MY</td>
<td>2.380</td>
</tr>
<tr>
<td>45</td>
<td>FX</td>
<td>3.609</td>
<td>MY</td>
<td>1.587</td>
</tr>
<tr>
<td>46</td>
<td>FX</td>
<td>3.609</td>
<td>MY</td>
<td>1.587</td>
</tr>
<tr>
<td>80</td>
<td>FX</td>
<td>3.609</td>
<td>MY</td>
<td>1.587</td>
</tr>
<tr>
<td>81</td>
<td>FX</td>
<td>5.413</td>
<td>MY</td>
<td>2.380</td>
</tr>
<tr>
<td>82</td>
<td>FX</td>
<td>3.609</td>
<td>MY</td>
<td>1.587</td>
</tr>
<tr>
<td></td>
<td></td>
<td>-------</td>
<td>----</td>
<td>-------</td>
</tr>
<tr>
<td></td>
<td>TOTAL = 82.998</td>
<td>36.493 AT LEVEL 6.000 METE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>49</td>
<td>FX</td>
<td>9.357</td>
<td>MY</td>
<td>4.371</td>
</tr>
<tr>
<td>50</td>
<td>FX</td>
<td>9.357</td>
<td>MY</td>
<td>4.371</td>
</tr>
<tr>
<td>51</td>
<td>FX</td>
<td>12.477</td>
<td>MY</td>
<td>5.829</td>
</tr>
<tr>
<td>52</td>
<td>FX</td>
<td>12.477</td>
<td>MY</td>
<td>5.829</td>
</tr>
<tr>
<td>53</td>
<td>FX</td>
<td>12.477</td>
<td>MY</td>
<td>5.829</td>
</tr>
<tr>
<td>54</td>
<td>FX</td>
<td>12.477</td>
<td>MY</td>
<td>5.829</td>
</tr>
<tr>
<td>55</td>
<td>FX</td>
<td>6.238</td>
<td>MY</td>
<td>2.914</td>
</tr>
<tr>
<td>56</td>
<td>FX</td>
<td>6.238</td>
<td>MY</td>
<td>2.914</td>
</tr>
<tr>
<td>57</td>
<td>FX</td>
<td>6.238</td>
<td>MY</td>
<td>2.914</td>
</tr>
<tr>
<td>58</td>
<td>FX</td>
<td>12.477</td>
<td>MY</td>
<td>5.829</td>
</tr>
<tr>
<td>59</td>
<td>FX</td>
<td>9.357</td>
<td>MY</td>
<td>4.371</td>
</tr>
<tr>
<td>70</td>
<td>FX</td>
<td>6.238</td>
<td>MY</td>
<td>2.914</td>
</tr>
<tr>
<td>71</td>
<td>FX</td>
<td>6.238</td>
<td>MY</td>
<td>2.914</td>
</tr>
<tr>
<td>84</td>
<td>FX</td>
<td>6.238</td>
<td>MY</td>
<td>2.914</td>
</tr>
<tr>
<td>85</td>
<td>FX</td>
<td>9.357</td>
<td>MY</td>
<td>4.371</td>
</tr>
<tr>
<td>86</td>
<td>FX</td>
<td>6.238</td>
<td>MY</td>
<td>2.914</td>
</tr>
<tr>
<td></td>
<td></td>
<td>-------</td>
<td>----</td>
<td>-------</td>
</tr>
<tr>
<td></td>
<td>TOTAL = 143.480</td>
<td>67.029 AT LEVEL 9.000 METE</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

************** END OF DATA FROM INTERNAL STORAGE **************

76. PRINT DIA CR

DIA CR

*******************************************************************************
Related Links

- **TR.31.2.15 IBC 2018 Seismic Load Definition** (on page 2603)

### V.IBC 2018 Static Seismic T Greater Than 2.5

Verify the program-calculated base shear and its distribution along the height of a 22-story frame by using equivalent lateral force method as per IBC 2018. Also, verify the torsional moments to which the floors are subjected to, considering inherent as well as accidental torsion.

**Details**

A 22-story structure is subject to a seismic load from the +X direction (see following figure). Each story is 10' in height (total building height of 220’). The column bases are assumed fixed. The time period of the structure uses is its Rayleigh frequency.

![Figure 411: Plan View](image-url)
Assumptions

i. Mapped MCER spectral response acceleration parameter at short period, $S_s = 1.5$

ii. Mapped MCER spectral response acceleration parameter at a period of 1 s, $S_1 = 0.5$

iii. Risk Category – I (Hence, From Table 1.5-2, Importance Factor $I = 1$)

iv. Site Class – D (SCLASS 4)

v. Response Modification Factor ($R_X$ & $R_Z$) = 3

vi. Long-period Transition Time ($T_L$) = 12 s

vii. Seismic weight is composed of UDLs (magnitude -22.481 kip/ft, direction GY) (defined in Reference Load Definition), incident on the beams

viii. The effect of shear deformation is neglected

Validation

Calculation of Base Shear

Based on $S_s$ and $S_1$, $F_a = 1$ (per table 11.4-1) and $F_v = 1.8$ (per table 11.4-2).

Hence:

$$ S_{MS} = F_a \times S_s = 1 \times 1.5 = 1.5 $$

Eq. 11.4-1

$$ S_{DS} = \frac{2}{3} \times S_{MS} = 1.0 $$

Eq. 11.4-3

$$ S_{M1} = F_v \times S_1 = 1.8 \times 0.5 = 0.9 $$

Eq. 11.4-3

$$ S_{D1} = \frac{2}{3} \times S_{M1} = 0.60 $$

Eq. 11.4-4

$$ (C_s)_{initial} = \frac{S_{DS}}{R/T} = 0.333 $$

Eq. 12.8-2

The time period of the structure is taken as the Rayleigh frequency of the structure (taken from the output file), $T_R = 18.12$ s.

Height of the structure, $h = 220$ ft.

For steel moment-resisting frames, $C_t = 0.028$ and $x = 0.8$ (per Table 12.8-2).

So the approximate time period, $T_a = C_t \times h^x = 0.028(220 \text{ ft})^{0.8} = 2.0946$ s

From Table 12.8-1, $C_u = 1.4$ (for $S_{D1} > 0.4$).

$$ C_u \times T_a = 2.9324 \text{ s} $$

The time period used, $T$, is the lesser of $T_R$ and $C_u \times T_a$, which is $2.9324$ s ($< T_L = 12$ s).

$$ (C_s)_{max} = \frac{S_{D1}}{T \times (R/T)} = 0.0682 $$

Eq. 12.8-3

$$ (C_s)_{min1} = 0.044 \times S_{DS} \times I = 0.044 $$

Eq. 12.8-5

Since $S_1 = 0.5 < 0.6g$, equation 12.8-6 does not need to be considered for calculating the lower limit of $C_s$:

Therefore, $C_s = 0.0682$.

The seismic weight, $W$ is taken as the total seismic weight of the beams: $= 396 \times 22.481 \times 10 = 89,024.77$ kip

So the total base shear, $V = C_s \times W = 0.0682 \times 89,024.77 = 6,071.50$ kip

Vertical Distribution of Lateral Forces

Since, time period of the structure $T > 2.5$ s, as per clause 12.8.3, $k = 2$.  


Hence, from equation 12.8-11 and equation 12.8-12, we can find the lateral forces in different story levels, as follows:

**Table 439: Vertical force distribution**

<table>
<thead>
<tr>
<th>Story Level</th>
<th>$W_x$ (kip)</th>
<th>$h_x$ (ft)</th>
<th>$W_x h_x^k$</th>
<th>$\sum_{i=1}^{22} (W_i h_i^k)$</th>
<th>Lateral force at story level (kip)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4,046.58</td>
<td>10</td>
<td>404,658</td>
<td>0.00026</td>
<td>1.60</td>
</tr>
<tr>
<td>2</td>
<td>4,046.58</td>
<td>20</td>
<td>1,618,632</td>
<td>0.00105</td>
<td>6.40</td>
</tr>
<tr>
<td>3</td>
<td>4,046.58</td>
<td>30</td>
<td>3,641,922</td>
<td>0.00237</td>
<td>14.40</td>
</tr>
<tr>
<td>4</td>
<td>4,046.58</td>
<td>40</td>
<td>6,474,528</td>
<td>0.00422</td>
<td>25.60</td>
</tr>
<tr>
<td>5</td>
<td>4,046.58</td>
<td>50</td>
<td>10,116,450</td>
<td>0.00659</td>
<td>40.00</td>
</tr>
<tr>
<td>6</td>
<td>4,046.58</td>
<td>60</td>
<td>14,567,688</td>
<td>0.00949</td>
<td>57.60</td>
</tr>
<tr>
<td>7</td>
<td>4,046.58</td>
<td>70</td>
<td>19,828,242</td>
<td>0.01291</td>
<td>78.39</td>
</tr>
<tr>
<td>8</td>
<td>4,046.58</td>
<td>80</td>
<td>25,898,112</td>
<td>0.01686</td>
<td>102.39</td>
</tr>
<tr>
<td>9</td>
<td>4,046.58</td>
<td>90</td>
<td>32,777,298</td>
<td>0.02134</td>
<td>129.59</td>
</tr>
<tr>
<td>10</td>
<td>4,046.58</td>
<td>100</td>
<td>40,465,800</td>
<td>0.02635</td>
<td>159.99</td>
</tr>
<tr>
<td>11</td>
<td>4,046.58</td>
<td>110</td>
<td>48,963,618</td>
<td>0.03188</td>
<td>193.58</td>
</tr>
<tr>
<td>12</td>
<td>4,046.58</td>
<td>120</td>
<td>58,270,752</td>
<td>0.03794</td>
<td>230.38</td>
</tr>
<tr>
<td>13</td>
<td>4,046.58</td>
<td>130</td>
<td>68,387,202</td>
<td>0.04453</td>
<td>270.38</td>
</tr>
<tr>
<td>14</td>
<td>4,046.58</td>
<td>140</td>
<td>79,312,968</td>
<td>0.05165</td>
<td>313.57</td>
</tr>
<tr>
<td>15</td>
<td>4,046.58</td>
<td>150</td>
<td>91,048,050</td>
<td>0.05929</td>
<td>359.97</td>
</tr>
<tr>
<td>16</td>
<td>4,046.58</td>
<td>160</td>
<td>103,592,448</td>
<td>0.06746</td>
<td>409.57</td>
</tr>
<tr>
<td>17</td>
<td>4,046.58</td>
<td>170</td>
<td>116,946,162</td>
<td>0.07615</td>
<td>462.36</td>
</tr>
<tr>
<td>18</td>
<td>4,046.58</td>
<td>180</td>
<td>131,109,192</td>
<td>0.08538</td>
<td>518.36</td>
</tr>
<tr>
<td>19</td>
<td>4,046.58</td>
<td>190</td>
<td>146,081,538</td>
<td>0.09513</td>
<td>577.55</td>
</tr>
<tr>
<td>20</td>
<td>4,046.58</td>
<td>200</td>
<td>161,863,200</td>
<td>0.10540</td>
<td>639.95</td>
</tr>
<tr>
<td>21</td>
<td>4,046.58</td>
<td>210</td>
<td>178,454,178</td>
<td>0.11621</td>
<td>705.54</td>
</tr>
<tr>
<td>22</td>
<td>4,046.58</td>
<td>220</td>
<td>195,854,472</td>
<td>0.12754</td>
<td>774.33</td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.06 Loading

<table>
<thead>
<tr>
<th>Story Level</th>
<th>$W_x$ (kip)</th>
<th>$h_x$ (ft)</th>
<th>$W_x \times h_x$</th>
<th>$\sum_{i=1}^{n} (W_i \times h_i)$</th>
<th>Lateral force at story level (kip)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Sigma$</td>
<td>89,024.76</td>
<td></td>
<td>1,535,677,110</td>
<td>1</td>
<td>6071.50</td>
</tr>
</tbody>
</table>

**Consideration of Inherent and Accidental Torsion (as per Clause 12.8.4.1 and 12.8.4.2)**

From the output, we can find the Center of Mass and Center of Rigidity values for different story of the structure, numerical difference between the two gives us the static eccentricity. For dynamic eccentricity calculation, the static eccentricity values are to be multiplied with a factor ($DEC - 1$) ($DEC$ being the dynamic eccentricity factor, provided in the seismic load item in the model = 2), so ($DEC-1$) = 1. For accidental eccentricity, the multiplying factor $ACC$ ($ACC$ being the multiplication factor for Accidental Torsional Moment, provided in the seismic load item in the model = 0.05). These two eccentricities add up to give the total eccentricity due to inherent as well as accidental torsion. This eccentricity multiplied by the Lateral Force in a particular story level, gives the total torsional moment in that level. Calculations show the results to be:

<table>
<thead>
<tr>
<th>Floor Level</th>
<th>Center of Mass, $CM$ (ft)</th>
<th>Center of Rigidity CR (ft)</th>
<th>Static Eccentricity $SE = CM-CR$ (ft)</th>
<th>Dynamic Eccentricity $DE = SE \times DEC$ (ft)</th>
<th>Accidental Torsional Eccentricity $AE$ (ft)</th>
<th>$DE+AE$ (ft)</th>
<th>Lateral Force in the Story (kip)</th>
<th>Torsional Moment (kip-ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>13.611</td>
<td>13.76</td>
<td>-0.149</td>
<td>-0.149</td>
<td>0.05 x 30 = 1.5</td>
<td>1.351</td>
<td>1.6</td>
<td>2.162</td>
</tr>
<tr>
<td>2</td>
<td>13.611</td>
<td>13.643</td>
<td>-0.032</td>
<td>-0.032</td>
<td>1.5</td>
<td>1.468</td>
<td>6.4</td>
<td>9.395</td>
</tr>
<tr>
<td>3</td>
<td>13.611</td>
<td>13.575</td>
<td>0.036</td>
<td>0.036</td>
<td>1.5</td>
<td>1.536</td>
<td>14.4</td>
<td>22.12</td>
</tr>
<tr>
<td>4</td>
<td>13.611</td>
<td>13.524</td>
<td>0.087</td>
<td>0.087</td>
<td>1.5</td>
<td>1.587</td>
<td>25.6</td>
<td>40.63</td>
</tr>
<tr>
<td>5</td>
<td>13.611</td>
<td>13.481</td>
<td>0.13</td>
<td>0.13</td>
<td>1.5</td>
<td>1.63</td>
<td>40.0</td>
<td>65.20</td>
</tr>
<tr>
<td>6</td>
<td>13.611</td>
<td>13.443</td>
<td>0.168</td>
<td>0.168</td>
<td>1.5</td>
<td>1.668</td>
<td>57.6</td>
<td>96.07</td>
</tr>
<tr>
<td>7</td>
<td>13.611</td>
<td>13.408</td>
<td>0.203</td>
<td>0.203</td>
<td>1.5</td>
<td>1.703</td>
<td>78.4</td>
<td>133.5</td>
</tr>
<tr>
<td>8</td>
<td>13.611</td>
<td>13.378</td>
<td>0.233</td>
<td>0.233</td>
<td>1.5</td>
<td>1.733</td>
<td>102.4</td>
<td>177.5</td>
</tr>
<tr>
<td>9</td>
<td>13.611</td>
<td>13.350</td>
<td>0.261</td>
<td>0.261</td>
<td>1.5</td>
<td>1.761</td>
<td>129.6</td>
<td>228.2</td>
</tr>
<tr>
<td>10</td>
<td>13.611</td>
<td>13.326</td>
<td>0.285</td>
<td>0.285</td>
<td>1.5</td>
<td>1.785</td>
<td>156.0</td>
<td>285.6</td>
</tr>
<tr>
<td>11</td>
<td>13.611</td>
<td>13.305</td>
<td>0.306</td>
<td>0.306</td>
<td>1.5</td>
<td>1.806</td>
<td>193.6</td>
<td>349.6</td>
</tr>
<tr>
<td>12</td>
<td>13.611</td>
<td>13.286</td>
<td>0.325</td>
<td>0.325</td>
<td>1.5</td>
<td>1.825</td>
<td>230.4</td>
<td>420.5</td>
</tr>
<tr>
<td>13</td>
<td>13.611</td>
<td>13.269</td>
<td>0.342</td>
<td>0.342</td>
<td>1.5</td>
<td>1.842</td>
<td>270.4</td>
<td>498.1</td>
</tr>
</tbody>
</table>
Results

Table 440: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base shear, V (kN)</td>
<td>6,071.50</td>
<td>6,071.79</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Lateral force (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1st Floor</td>
<td>1.6</td>
<td>1.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>2nd Floor</td>
<td>6.4</td>
<td>6.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>3rd Floor</td>
<td>14.4</td>
<td>14.399</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>4th Floor</td>
<td>25.6</td>
<td>25.599</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>5th Floor</td>
<td>40.0</td>
<td>39.999</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>6th Floor</td>
<td>57.6</td>
<td>57.598</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>7th Floor</td>
<td>78.4</td>
<td>78.397</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>8th Floor</td>
<td>102.4</td>
<td>102.396</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>9th Floor</td>
<td>129.6</td>
<td>129.595</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Result Type</td>
<td>Reference</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>---------------------</td>
<td>-----------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>10th Floor</td>
<td>160.0</td>
<td>159.994</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>11th Floor</td>
<td>193.6</td>
<td>193.593</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>12th Floor</td>
<td>230.4</td>
<td>230.392</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>13th Floor</td>
<td>270.4</td>
<td>270.391</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>14th Floor</td>
<td>313.6</td>
<td>313.589</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>15th Floor</td>
<td>360.0</td>
<td>359.987</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>16th Floor</td>
<td>409.6</td>
<td>409.586</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>17th Floor</td>
<td>462.4</td>
<td>462.384</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>18th Floor</td>
<td>518.4</td>
<td>518.382</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>19th Floor</td>
<td>577.6</td>
<td>577.580</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>20th Floor</td>
<td>640.0</td>
<td>639.978</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>21st Floor</td>
<td>705.6</td>
<td>705.575</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>22nd Floor</td>
<td>774.4</td>
<td>774.373</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Torsional moment (kN·m)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1st Floor</td>
<td>2.162</td>
<td>2.162</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>2nd Floor</td>
<td>9.395</td>
<td>9.397</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>3rd Floor</td>
<td>22.12</td>
<td>22.124</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>4th Floor</td>
<td>40.63</td>
<td>40.626</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>5th Floor</td>
<td>65.20</td>
<td>65.206</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>6th Floor</td>
<td>96.07</td>
<td>96.105</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>7th Floor</td>
<td>133.5</td>
<td>135.504</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>8th Floor</td>
<td>177.5</td>
<td>177.510</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>9th Floor</td>
<td>228.2</td>
<td>228.201</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>10th Floor</td>
<td>285.6</td>
<td>285.605</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>11th Floor</td>
<td>349.6</td>
<td>349.729</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>12th Floor</td>
<td>420.5</td>
<td>420.564</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>13th Floor</td>
<td>498.1</td>
<td>498.099</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Result Type</td>
<td>Reference</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>-------------</td>
<td>-----------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>14th Floor</td>
<td>582.3</td>
<td>582.330</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>15th Floor</td>
<td>673.2</td>
<td>673.273</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>16th Floor</td>
<td>770.8</td>
<td>770.91</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>17th Floor</td>
<td>875.8</td>
<td>875.615</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>18th Floor</td>
<td>987.5</td>
<td>987.393</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>19th Floor</td>
<td>1,106.6</td>
<td>1,106.747</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>20th Floor</td>
<td>1,234.5</td>
<td>1,234.386</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>21st Floor</td>
<td>1,371.6</td>
<td>1,371.456</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>22nd Floor</td>
<td>1,521.6</td>
<td>1,521.448</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\2018\IBC 2018 Static Seismic T Greater Than 2.5.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Mar-19
END JOB INFORMATION
*****************************************************************************
*This problem is created to verify the base shear, distribution of base shear
*And Inherent and Accidental Torsional Moment at different floor levels
*Of the Structure
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10; 9 0 0 20; 10 0 10 20; 11 10 10 20; 12 10 0 20; 13 20 10 0;
14 20 0 0; 15 20 10 10; 16 20 0 10; 17 20 10 20; 18 20 0 20; 21 30 10 10;
22 30 0 10; 23 30 10 20; 24 30 0 20; 33 10 10 30; 34 10 0 30; 35 20 10 30;
36 20 0 30; 37 0 20 0; 38 10 20 0; 39 0 20 10; 40 10 20 10; 41 0 20 20;
42 10 20 20; 43 20 20 0; 44 20 20 10; 45 20 20 20; 46 30 20 10; 47 30 20 20;
48 10 20 30; 49 20 20 30; 50 0 30 0; 51 10 30 0; 52 0 30 10; 53 10 30 10;
54 0 30 20; 55 10 30 20; 56 20 30 0; 57 20 30 10; 58 20 30 20; 59 30 30 10;
60 30 30 20; 61 10 30 30; 62 20 30 30; 63 0 40 0; 64 10 40 0; 65 0 40 10;
66 10 40 10; 67 0 40 20; 68 10 40 20; 69 20 40 0; 70 20 40 10; 71 20 40 20;
72 30 40 10; 73 30 40 20; 74 10 40 30; 75 20 40 30; 76 0 50 0; 77 10 50 0;
78 0 50 10; 79 10 50 10; 80 0 50 20; 81 10 50 20; 82 20 50 0; 83 20 50 10;
84 20 50 20; 85 30 50 10; 86 30 50 20; 87 10 50 30; 88 20 50 30; 89 0 60 0;
90 10 60 0; 91 0 60 10; 92 10 60 10; 93 0 60 20; 94 10 60 20; 95 20 60 0;
96 20 60 10; 97 20 60 20; 98 30 60 10; 99 30 60 20; 100 10 60 30; 101 20 60 30;
Verification Examples

V.06 Loading

STAAD.Pro

MEMBER INCIDENCES

User Manual
Verification Examples
V.06 Loading
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 21 23 24 27 TO 29 39 40 45 TO 698 TABLE ST W12X58
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 5 8 9 12 14 16 18 22 24 34 36 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
MEMBER LOAD
  2 4 5 7 9 10 12 14 TO 16 18 20 23 24 28 39 40 46 61 TO 78 92 TO 109 -
  123 TO 140 154 TO 171 185 TO 202 216 TO 233 247 TO 264 278 TO 295 -
  309 TO 326 340 TO 357 371 TO 388 402 TO 419 433 TO 450 464 TO 481 -
  495 TO 512 526 TO 543 557 TO 574 588 TO 605 619 TO 636 650 TO 667 -
  681 TO 698 UNI GY -22.481
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 10
DIA 2 TYPE RIG HEI 20
DIA 3 TYPE RIG HEI 30
DIA 4 TYPE RIG HEI 40
DIA 5 TYPE RIG HEI 50
DIA 6 TYPE RIG HEI 60
DIA 7 TYPE RIG HEI 70
DIA 8 TYPE RIG HEI 80
DIA 9 TYPE RIG HEI 90
DIA 10 TYPE RIG HEI 100
DIA 11 TYPE RIG HEI 110
DIA 12 TYPE RIG HEI 120
DIA 13 TYPE RIG HEI 130
DIA 14 TYPE RIG HEI 140
DIA 15 TYPE RIG HEI 150
DIA 16 TYPE RIG HEI 160
DIA 17 TYPE RIG HEI 170
DIA 18 TYPE RIG HEI 180
DIA 19 TYPE RIG HEI 190
DIA 20 TYPE RIG HEI 200
DIA 21 TYPE RIG HEI 210
DIA 22 TYPE RIG HEI 220
DEFINE IBC 2018
SS 1.5 S1 0.5 I 1 RX 3 RZ 3 SCLASS 4 TL 12
LOAD 1 LOADTYPE Seismic TITLE SL +X
IBC LOAD X 1 DEC 2 ACC 0.05
PERFORM ANALYSIS PRINT LOAD DATA
PRI DIA CR
FINISH

STAAD.Pro Output

*********************************************************************************************
* EQUIV. SEISMIC LOADS AS PER IBC                                                   *
* PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD                                           *
* SS =  1.500 S1 =  0.500 FA =  1.000 FV = 1.800                                     *
* SDS =  1.000 SD1 = 0.600                                                         *
*********************************************************************************************
230. LOAD 1 LOADTYPE SEISMIC TITLE SL +X
231. IBC LOAD X 1 DEC 2 ACC 0.05
232. PERFORM ANALYSIS PRINT LOAD DATA

STAAD.Pro 3460 User Manual
### Problem Statistics

- **Number of Joints**: 321
- **Number of Members**: 682
- **Number of Plates**: 0
- **Number of Solids**: 0
- **Number of Surfaces**: 0
- **Number of Supports**: 13

Using 64-bit analysis engine. 

SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 990

TOTAL LOAD COMBINATION CASES = 0 so far.

---

### Loading 1

**Load Type**: Seismic  
**Title**: SL +X

**IBC 2018 Seismic Load Along X**:  
CT = 0.028  
Cu = 1.400  
x = 0.8000

**Time Periods**:  
Ta = 2.095  
T = 18.120  
Tuser = 0.000

**Cs Limits**:  
Lower = 0.044  
Upper = 0.068

**Load Factor**: 1.000

**Design Base Shear**:  
1.000 X 0.068 X 89024.77 = 6071.79 KIP

---

***Note: Seismic load is acting at center of mass for rigid diaphragm. Torsion from static eccentricity (esi) is included in analysis. Dynamic eccentricity applied = DEC - 1***

**Load No.: 1**  
**Direction**: X  
**Unit**: Feet

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
<td>Z</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td>dec</td>
<td>+aec</td>
<td>dec</td>
</tr>
<tr>
<td>1</td>
<td>10.00</td>
<td>-0.22</td>
<td>-0.15</td>
</tr>
<tr>
<td>2</td>
<td>20.00</td>
<td>-0.19</td>
<td>-0.03</td>
</tr>
<tr>
<td>3</td>
<td>30.00</td>
<td>-0.17</td>
<td>0.04</td>
</tr>
<tr>
<td>4</td>
<td>40.00</td>
<td>-0.15</td>
<td>0.09</td>
</tr>
<tr>
<td>5</td>
<td>50.00</td>
<td>-0.14</td>
<td>0.13</td>
</tr>
<tr>
<td>6</td>
<td>60.00</td>
<td>-0.12</td>
<td>0.17</td>
</tr>
<tr>
<td>7</td>
<td>70.00</td>
<td>-0.11</td>
<td>0.20</td>
</tr>
<tr>
<td>8</td>
<td>80.00</td>
<td>-0.10</td>
<td>0.23</td>
</tr>
<tr>
<td>9</td>
<td>90.00</td>
<td>-0.09</td>
<td>0.26</td>
</tr>
<tr>
<td>10</td>
<td>100.00</td>
<td>-0.08</td>
<td>0.29</td>
</tr>
<tr>
<td>11</td>
<td>110.00</td>
<td>-0.07</td>
<td>0.31</td>
</tr>
<tr>
<td>12</td>
<td>120.00</td>
<td>-0.06</td>
<td>0.33</td>
</tr>
<tr>
<td>13</td>
<td>130.00</td>
<td>-0.05</td>
<td>0.34</td>
</tr>
<tr>
<td>14</td>
<td>140.00</td>
<td>-0.04</td>
<td>0.36</td>
</tr>
<tr>
<td>15</td>
<td>150.00</td>
<td>-0.04</td>
<td>0.37</td>
</tr>
<tr>
<td>16</td>
<td>160.00</td>
<td>-0.03</td>
<td>0.38</td>
</tr>
<tr>
<td>17</td>
<td>170.00</td>
<td>-0.02</td>
<td>0.39</td>
</tr>
<tr>
<td>18</td>
<td>180.00</td>
<td>-0.02</td>
<td>0.40</td>
</tr>
<tr>
<td>19</td>
<td>190.00</td>
<td>-0.01</td>
<td>0.42</td>
</tr>
<tr>
<td>20</td>
<td>200.00</td>
<td>-0.01</td>
<td>0.43</td>
</tr>
<tr>
<td>21</td>
<td>210.00</td>
<td>0.00</td>
<td>0.44</td>
</tr>
<tr>
<td>JOINT</td>
<td>LATERAL LOAD (KIP)</td>
<td>TORSIONAL MOMENT (KIP- FEET)</td>
<td>LOAD - 1 FACTOR - 1.000</td>
</tr>
<tr>
<td>-------</td>
<td>-------------------</td>
<td>-----------------------------</td>
<td>-------------------------</td>
</tr>
<tr>
<td>DEC + AEC</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>FX 0.089</td>
<td>MY 0.120</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>FX 0.133</td>
<td>MY 0.180</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>FX 0.133</td>
<td>MY 0.180</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>FX 0.178</td>
<td>MY 0.240</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>FX 0.089</td>
<td>MY 0.120</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>FX 0.178</td>
<td>MY 0.240</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>FX 0.089</td>
<td>MY 0.120</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>FX 0.178</td>
<td>MY 0.240</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>FX 0.178</td>
<td>MY 0.240</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>FX 0.133</td>
<td>MY 0.180</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>FX 0.089</td>
<td>MY 0.120</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>FX 0.089</td>
<td>MY 0.120</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>FX 0.089</td>
<td>MY 0.120</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL = 1.600</td>
<td>2.162 AT LEVEL 10.000 FEET</td>
<td></td>
<td></td>
</tr>
<tr>
<td>37</td>
<td>FX 0.356</td>
<td>MY 0.522</td>
<td></td>
</tr>
<tr>
<td>38</td>
<td>FX 0.533</td>
<td>MY 0.783</td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>FX 0.533</td>
<td>MY 0.783</td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>FX 0.711</td>
<td>MY 1.044</td>
<td></td>
</tr>
<tr>
<td>41</td>
<td>FX 0.356</td>
<td>MY 0.522</td>
<td></td>
</tr>
<tr>
<td>42</td>
<td>FX 0.711</td>
<td>MY 1.044</td>
<td></td>
</tr>
<tr>
<td>43</td>
<td>FX 0.356</td>
<td>MY 0.522</td>
<td></td>
</tr>
<tr>
<td>44</td>
<td>FX 0.711</td>
<td>MY 1.044</td>
<td></td>
</tr>
<tr>
<td>45</td>
<td>FX 0.711</td>
<td>MY 1.044</td>
<td></td>
</tr>
<tr>
<td>46</td>
<td>FX 0.356</td>
<td>MY 0.522</td>
<td></td>
</tr>
<tr>
<td>47</td>
<td>FX 0.356</td>
<td>MY 0.522</td>
<td></td>
</tr>
<tr>
<td>48</td>
<td>FX 0.356</td>
<td>MY 0.522</td>
<td></td>
</tr>
<tr>
<td>49</td>
<td>FX 0.356</td>
<td>MY 0.522</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL = 6.400</td>
<td>9.397 AT LEVEL 20.000 FEET</td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>FX 0.800</td>
<td>MY 1.229</td>
<td></td>
</tr>
<tr>
<td>51</td>
<td>FX 1.200</td>
<td>MY 1.844</td>
<td></td>
</tr>
<tr>
<td>52</td>
<td>FX 1.200</td>
<td>MY 1.844</td>
<td></td>
</tr>
<tr>
<td>53</td>
<td>FX 1.600</td>
<td>MY 2.458</td>
<td></td>
</tr>
<tr>
<td>54</td>
<td>FX 0.800</td>
<td>MY 1.229</td>
<td></td>
</tr>
<tr>
<td>STAAD SPACE                         -- PAGE NO.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>55</td>
<td>FX 1.600</td>
<td>MY 2.458</td>
<td></td>
</tr>
<tr>
<td>56</td>
<td>FX 0.800</td>
<td>MY 1.229</td>
<td></td>
</tr>
<tr>
<td>57</td>
<td>FX 1.600</td>
<td>MY 2.458</td>
<td></td>
</tr>
<tr>
<td>58</td>
<td>FX 1.600</td>
<td>MY 2.458</td>
<td></td>
</tr>
<tr>
<td>59</td>
<td>FX 0.800</td>
<td>MY 1.229</td>
<td></td>
</tr>
<tr>
<td>60</td>
<td>FX 0.800</td>
<td>MY 1.229</td>
<td></td>
</tr>
<tr>
<td>61</td>
<td>FX 0.800</td>
<td>MY 1.229</td>
<td></td>
</tr>
<tr>
<td>62</td>
<td>FX 0.800</td>
<td>MY 1.229</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL = 14.399</td>
<td>22.124 AT LEVEL 30.000 FEET</td>
<td></td>
<td></td>
</tr>
<tr>
<td>63</td>
<td>FX 1.422</td>
<td>MY 2.257</td>
<td></td>
</tr>
<tr>
<td>64</td>
<td>FX 2.133</td>
<td>MY 3.385</td>
<td></td>
</tr>
<tr>
<td>65</td>
<td>FX 2.133</td>
<td>MY 3.385</td>
<td></td>
</tr>
<tr>
<td>66</td>
<td>FX 2.844</td>
<td>MY 4.514</td>
<td></td>
</tr>
<tr>
<td>67</td>
<td>FX 1.422</td>
<td>MY 2.257</td>
<td></td>
</tr>
</tbody>
</table>

**Verification Examples**

**V.06 Loading**

User Manual
<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th></th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>68</td>
<td>2.844</td>
<td>4.514</td>
<td></td>
<td></td>
</tr>
<tr>
<td>69</td>
<td>1.422</td>
<td>2.257</td>
<td></td>
<td></td>
</tr>
<tr>
<td>70</td>
<td>2.844</td>
<td>4.514</td>
<td></td>
<td></td>
</tr>
<tr>
<td>71</td>
<td>2.844</td>
<td>4.514</td>
<td></td>
<td></td>
</tr>
<tr>
<td>72</td>
<td>1.422</td>
<td>2.257</td>
<td></td>
<td></td>
</tr>
<tr>
<td>73</td>
<td>1.422</td>
<td>2.257</td>
<td></td>
<td></td>
</tr>
<tr>
<td>74</td>
<td>1.422</td>
<td>2.257</td>
<td></td>
<td></td>
</tr>
<tr>
<td>75</td>
<td>1.422</td>
<td>2.257</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL = 25.599</td>
<td>40.626 AT LEVEL 40.000 FEET</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>76</td>
<td>2.222</td>
<td>3.623</td>
<td></td>
<td></td>
</tr>
<tr>
<td>77</td>
<td>3.333</td>
<td>5.434</td>
<td></td>
<td></td>
</tr>
<tr>
<td>78</td>
<td>3.333</td>
<td>5.434</td>
<td></td>
<td></td>
</tr>
<tr>
<td>79</td>
<td>4.444</td>
<td>7.245</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80</td>
<td>2.222</td>
<td>3.623</td>
<td></td>
<td></td>
</tr>
<tr>
<td>81</td>
<td>4.444</td>
<td>7.245</td>
<td></td>
<td></td>
</tr>
<tr>
<td>82</td>
<td>2.222</td>
<td>3.623</td>
<td></td>
<td></td>
</tr>
<tr>
<td>83</td>
<td>4.444</td>
<td>7.245</td>
<td></td>
<td></td>
</tr>
<tr>
<td>84</td>
<td>4.444</td>
<td>7.245</td>
<td></td>
<td></td>
</tr>
<tr>
<td>85</td>
<td>2.222</td>
<td>3.623</td>
<td></td>
<td></td>
</tr>
<tr>
<td>86</td>
<td>2.222</td>
<td>3.623</td>
<td></td>
<td></td>
</tr>
<tr>
<td>87</td>
<td>2.222</td>
<td>3.623</td>
<td></td>
<td></td>
</tr>
<tr>
<td>88</td>
<td>2.222</td>
<td>3.623</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL = 39.999</td>
<td>65.206 AT LEVEL 50.000 FEET</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>89</td>
<td>3.200</td>
<td>5.339</td>
<td></td>
<td></td>
</tr>
<tr>
<td>90</td>
<td>4.800</td>
<td>8.009</td>
<td></td>
<td></td>
</tr>
<tr>
<td>91</td>
<td>4.800</td>
<td>8.009</td>
<td></td>
<td></td>
</tr>
<tr>
<td>92</td>
<td>6.400</td>
<td>10.678</td>
<td></td>
<td></td>
</tr>
<tr>
<td>93</td>
<td>3.200</td>
<td>5.339</td>
<td></td>
<td></td>
</tr>
<tr>
<td>94</td>
<td>6.400</td>
<td>10.678</td>
<td></td>
<td></td>
</tr>
<tr>
<td>95</td>
<td>3.200</td>
<td>5.339</td>
<td></td>
<td></td>
</tr>
<tr>
<td>96</td>
<td>6.400</td>
<td>10.678</td>
<td></td>
<td></td>
</tr>
<tr>
<td>97</td>
<td>6.400</td>
<td>10.678</td>
<td></td>
<td></td>
</tr>
<tr>
<td>98</td>
<td>3.200</td>
<td>5.339</td>
<td></td>
<td></td>
</tr>
<tr>
<td>99</td>
<td>3.200</td>
<td>5.339</td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>3.200</td>
<td>5.339</td>
<td></td>
<td></td>
</tr>
<tr>
<td>101</td>
<td>3.200</td>
<td>5.339</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL = 57.598</td>
<td>96.105 AT LEVEL 60.000 FEET</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>102</td>
<td>4.355</td>
<td>7.417</td>
<td></td>
<td></td>
</tr>
<tr>
<td>103</td>
<td>6.533</td>
<td>11.125</td>
<td></td>
<td></td>
</tr>
<tr>
<td>104</td>
<td>6.533</td>
<td>11.125</td>
<td></td>
<td></td>
</tr>
<tr>
<td>105</td>
<td>8.711</td>
<td>14.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>106</td>
<td>4.355</td>
<td>7.417</td>
<td></td>
<td></td>
</tr>
<tr>
<td>107</td>
<td>8.711</td>
<td>14.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>108</td>
<td>4.355</td>
<td>7.417</td>
<td></td>
<td></td>
</tr>
<tr>
<td>109</td>
<td>8.711</td>
<td>14.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>110</td>
<td>8.711</td>
<td>14.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>111</td>
<td>4.355</td>
<td>7.417</td>
<td></td>
<td></td>
</tr>
<tr>
<td>112</td>
<td>4.355</td>
<td>7.417</td>
<td></td>
<td></td>
</tr>
<tr>
<td>113</td>
<td>4.355</td>
<td>7.417</td>
<td></td>
<td></td>
</tr>
<tr>
<td>114</td>
<td>4.355</td>
<td>7.417</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL = 78.397</td>
<td>133.501 AT LEVEL 70.000 FEET</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>115</td>
<td>5.689</td>
<td>9.862</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Verification Examples**

V.06 Loading

|   | FX | MY |  
|---|----|----|---
| 116 | 8.533 | 14.793 |  
| 117 | 8.533 | 14.793 |  
| 118 | 11.377 | 19.723 |  
| 119 | 5.689 | 9.862 |  
| 120 | 11.377 | 19.723 |  
| 121 | 5.689 | 9.862 |  
| 122 | 11.377 | 19.723 |  
| 123 | 5.689 | 9.862 |  
| 124 | 5.689 | 9.862 |  
| 125 | 5.689 | 9.862 |  
| 126 | 5.689 | 9.862 |  
| 127 | 5.689 | 9.862 |  

|   | TOTAL | AT LEVEL | FEET |  
|---|-------|----------|------|---
|   | 102.396 | 80.000 |  
|   | 177.510 |  
|   | 129.595 | 90.000 |  
|   | 228.201 |  
|   | 159.994 | 100.000 |  
|   | 285.605 |  

STAAD SPACE

-- PAGE NO.
### Verification Examples

#### V.06 Loading

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th></th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>166</td>
<td>10.755</td>
<td>MY</td>
<td>19.429</td>
<td></td>
</tr>
<tr>
<td>167</td>
<td>12.800</td>
<td>MY</td>
<td>23.365</td>
<td></td>
</tr>
<tr>
<td>168</td>
<td>19.199</td>
<td>MY</td>
<td>35.047</td>
<td></td>
</tr>
<tr>
<td>169</td>
<td>19.199</td>
<td>MY</td>
<td>35.047</td>
<td></td>
</tr>
<tr>
<td>170</td>
<td>25.599</td>
<td>MY</td>
<td>46.729</td>
<td></td>
</tr>
<tr>
<td>171</td>
<td>12.800</td>
<td>MY</td>
<td>23.365</td>
<td></td>
</tr>
<tr>
<td>172</td>
<td>25.599</td>
<td>MY</td>
<td>46.729</td>
<td></td>
</tr>
<tr>
<td>173</td>
<td>25.599</td>
<td>MY</td>
<td>46.729</td>
<td></td>
</tr>
<tr>
<td>174</td>
<td>25.599</td>
<td>MY</td>
<td>46.729</td>
<td></td>
</tr>
<tr>
<td>175</td>
<td>12.800</td>
<td>MY</td>
<td>23.365</td>
<td></td>
</tr>
<tr>
<td>176</td>
<td>12.800</td>
<td>MY</td>
<td>23.365</td>
<td></td>
</tr>
<tr>
<td>177</td>
<td>12.800</td>
<td>MY</td>
<td>23.365</td>
<td></td>
</tr>
<tr>
<td>178</td>
<td>12.800</td>
<td>MY</td>
<td>23.365</td>
<td></td>
</tr>
<tr>
<td>179</td>
<td>12.800</td>
<td>MY</td>
<td>23.365</td>
<td></td>
</tr>
</tbody>
</table>

| TOTAL = 193.593 | 349.729 AT LEVEL 110.000 FEET |

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th></th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>180</td>
<td>15.022</td>
<td>MY</td>
<td>27.672</td>
<td></td>
</tr>
<tr>
<td>181</td>
<td>22.533</td>
<td>MY</td>
<td>41.508</td>
<td></td>
</tr>
<tr>
<td>182</td>
<td>22.533</td>
<td>MY</td>
<td>41.508</td>
<td></td>
</tr>
<tr>
<td>183</td>
<td>30.043</td>
<td>MY</td>
<td>55.344</td>
<td></td>
</tr>
<tr>
<td>184</td>
<td>15.022</td>
<td>MY</td>
<td>27.672</td>
<td></td>
</tr>
<tr>
<td>185</td>
<td>30.043</td>
<td>MY</td>
<td>55.344</td>
<td></td>
</tr>
<tr>
<td>186</td>
<td>15.022</td>
<td>MY</td>
<td>27.672</td>
<td></td>
</tr>
<tr>
<td>187</td>
<td>30.043</td>
<td>MY</td>
<td>55.344</td>
<td></td>
</tr>
<tr>
<td>188</td>
<td>30.043</td>
<td>MY</td>
<td>55.344</td>
<td></td>
</tr>
<tr>
<td>189</td>
<td>15.022</td>
<td>MY</td>
<td>27.672</td>
<td></td>
</tr>
<tr>
<td>190</td>
<td>15.022</td>
<td>MY</td>
<td>27.672</td>
<td></td>
</tr>
<tr>
<td>191</td>
<td>15.022</td>
<td>MY</td>
<td>27.672</td>
<td></td>
</tr>
<tr>
<td>192</td>
<td>15.022</td>
<td>MY</td>
<td>27.672</td>
<td></td>
</tr>
</tbody>
</table>

| TOTAL = 230.392 | 420.564 AT LEVEL 120.000 FEET |

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th></th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>193</td>
<td>17.422</td>
<td>MY</td>
<td>32.352</td>
<td></td>
</tr>
<tr>
<td>194</td>
<td>26.132</td>
<td>MY</td>
<td>48.527</td>
<td></td>
</tr>
<tr>
<td>195</td>
<td>26.132</td>
<td>MY</td>
<td>48.527</td>
<td></td>
</tr>
<tr>
<td>196</td>
<td>34.843</td>
<td>MY</td>
<td>64.703</td>
<td></td>
</tr>
<tr>
<td>197</td>
<td>17.422</td>
<td>MY</td>
<td>32.352</td>
<td></td>
</tr>
<tr>
<td>198</td>
<td>34.843</td>
<td>MY</td>
<td>64.703</td>
<td></td>
</tr>
<tr>
<td>199</td>
<td>17.422</td>
<td>MY</td>
<td>32.352</td>
<td></td>
</tr>
<tr>
<td>200</td>
<td>34.843</td>
<td>MY</td>
<td>64.703</td>
<td></td>
</tr>
<tr>
<td>201</td>
<td>34.843</td>
<td>MY</td>
<td>64.703</td>
<td></td>
</tr>
<tr>
<td>202</td>
<td>17.422</td>
<td>MY</td>
<td>32.352</td>
<td></td>
</tr>
<tr>
<td>203</td>
<td>17.422</td>
<td>MY</td>
<td>32.352</td>
<td></td>
</tr>
<tr>
<td>204</td>
<td>17.422</td>
<td>MY</td>
<td>32.352</td>
<td></td>
</tr>
<tr>
<td>205</td>
<td>17.422</td>
<td>MY</td>
<td>32.352</td>
<td></td>
</tr>
</tbody>
</table>

| TOTAL = 270.391 | 498.099 AT LEVEL 130.000 FEET |

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th></th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>206</td>
<td>19.999</td>
<td>MY</td>
<td>37.404</td>
<td></td>
</tr>
<tr>
<td>207</td>
<td>29.999</td>
<td>MY</td>
<td>56.106</td>
<td></td>
</tr>
<tr>
<td>208</td>
<td>29.999</td>
<td>MY</td>
<td>56.106</td>
<td></td>
</tr>
<tr>
<td>209</td>
<td>39.999</td>
<td>MY</td>
<td>74.808</td>
<td></td>
</tr>
<tr>
<td>210</td>
<td>19.999</td>
<td>MY</td>
<td>37.404</td>
<td></td>
</tr>
<tr>
<td>211</td>
<td>39.999</td>
<td>MY</td>
<td>74.808</td>
<td></td>
</tr>
<tr>
<td>212</td>
<td>19.999</td>
<td>MY</td>
<td>37.404</td>
<td></td>
</tr>
<tr>
<td>213</td>
<td>39.999</td>
<td>MY</td>
<td>74.808</td>
<td></td>
</tr>
</tbody>
</table>

| TOTAL = 313.589 | 582.330 AT LEVEL 140.000 FEET |
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Page</th>
<th>FX</th>
<th>MY</th>
<th>FX</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>214</td>
<td>39.999</td>
<td>74.808</td>
<td></td>
<td></td>
</tr>
<tr>
<td>215</td>
<td>19.999</td>
<td>37.404</td>
<td></td>
<td></td>
</tr>
<tr>
<td>216</td>
<td>19.999</td>
<td>37.404</td>
<td></td>
<td></td>
</tr>
<tr>
<td>217</td>
<td>19.999</td>
<td>37.404</td>
<td></td>
<td></td>
</tr>
<tr>
<td>218</td>
<td>19.999</td>
<td>37.404</td>
<td></td>
<td></td>
</tr>
<tr>
<td>219</td>
<td>22.755</td>
<td>42.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>220</td>
<td>34.132</td>
<td>64.249</td>
<td></td>
<td></td>
</tr>
<tr>
<td>221</td>
<td>34.132</td>
<td>64.249</td>
<td></td>
<td></td>
</tr>
<tr>
<td>222</td>
<td>45.510</td>
<td>85.666</td>
<td></td>
<td></td>
</tr>
<tr>
<td>223</td>
<td>22.755</td>
<td>42.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>224</td>
<td>45.510</td>
<td>85.666</td>
<td></td>
<td></td>
</tr>
<tr>
<td>225</td>
<td>22.755</td>
<td>42.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>226</td>
<td>45.510</td>
<td>85.666</td>
<td></td>
<td></td>
</tr>
<tr>
<td>227</td>
<td>45.510</td>
<td>85.666</td>
<td></td>
<td></td>
</tr>
<tr>
<td>228</td>
<td>22.755</td>
<td>42.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>229</td>
<td>22.755</td>
<td>42.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>230</td>
<td>22.755</td>
<td>42.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>231</td>
<td>22.755</td>
<td>42.833</td>
<td></td>
<td></td>
</tr>
<tr>
<td>232</td>
<td>25.688</td>
<td>48.645</td>
<td></td>
<td></td>
</tr>
<tr>
<td>233</td>
<td>38.532</td>
<td>72.968</td>
<td></td>
<td></td>
</tr>
<tr>
<td>234</td>
<td>38.532</td>
<td>72.968</td>
<td></td>
<td></td>
</tr>
<tr>
<td>235</td>
<td>51.376</td>
<td>97.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>236</td>
<td>25.688</td>
<td>48.645</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**STAAD SPACE**

---

**Page No. 14**

<table>
<thead>
<tr>
<th>PDF</th>
<th>FX</th>
<th>MY</th>
<th>FX</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>237</td>
<td>51.376</td>
<td>97.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>238</td>
<td>25.688</td>
<td>48.645</td>
<td></td>
<td></td>
</tr>
<tr>
<td>239</td>
<td>51.376</td>
<td>97.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>240</td>
<td>51.376</td>
<td>97.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>241</td>
<td>25.688</td>
<td>48.645</td>
<td></td>
<td></td>
</tr>
<tr>
<td>242</td>
<td>25.688</td>
<td>48.645</td>
<td></td>
<td></td>
</tr>
<tr>
<td>243</td>
<td>25.688</td>
<td>48.645</td>
<td></td>
<td></td>
</tr>
<tr>
<td>244</td>
<td>25.688</td>
<td>48.645</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**TOTAL = 462.384 AT LEVEL 170.000 FEET**

<table>
<thead>
<tr>
<th>PDF</th>
<th>FX</th>
<th>MY</th>
<th>FX</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>245</td>
<td>28.799</td>
<td>54.855</td>
<td></td>
<td></td>
</tr>
<tr>
<td>246</td>
<td>43.198</td>
<td>82.283</td>
<td></td>
<td></td>
</tr>
<tr>
<td>247</td>
<td>43.198</td>
<td>82.283</td>
<td></td>
<td></td>
</tr>
<tr>
<td>248</td>
<td>57.598</td>
<td>109.710</td>
<td></td>
<td></td>
</tr>
<tr>
<td>249</td>
<td>28.799</td>
<td>54.855</td>
<td></td>
<td></td>
</tr>
<tr>
<td>250</td>
<td>57.598</td>
<td>109.710</td>
<td></td>
<td></td>
</tr>
<tr>
<td>251</td>
<td>28.799</td>
<td>54.855</td>
<td></td>
<td></td>
</tr>
<tr>
<td>252</td>
<td>57.598</td>
<td>109.710</td>
<td></td>
<td></td>
</tr>
<tr>
<td>253</td>
<td>57.598</td>
<td>109.710</td>
<td></td>
<td></td>
</tr>
<tr>
<td>254</td>
<td>28.799</td>
<td>54.855</td>
<td></td>
<td></td>
</tr>
<tr>
<td>255</td>
<td>28.799</td>
<td>54.855</td>
<td></td>
<td></td>
</tr>
<tr>
<td>256</td>
<td>28.799</td>
<td>54.855</td>
<td></td>
<td></td>
</tr>
<tr>
<td>257</td>
<td>28.799</td>
<td>54.855</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**TOTAL = 518.382 AT LEVEL 180.000 FEET**

<table>
<thead>
<tr>
<th>PDF</th>
<th>FX</th>
<th>MY</th>
<th>FX</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>258</td>
<td>32.088</td>
<td>61.486</td>
<td></td>
<td></td>
</tr>
<tr>
<td>259</td>
<td>48.132</td>
<td>92.229</td>
<td></td>
<td></td>
</tr>
<tr>
<td>260</td>
<td>48.132</td>
<td>92.229</td>
<td></td>
<td></td>
</tr>
<tr>
<td>261</td>
<td>64.176</td>
<td>122.972</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

V.06 Loading

<table>
<thead>
<tr>
<th>262</th>
<th>FX</th>
<th>32.088</th>
<th>MY</th>
<th>61.486</th>
</tr>
</thead>
<tbody>
<tr>
<td>263</td>
<td>FX</td>
<td>64.176</td>
<td>MY</td>
<td>122.972</td>
</tr>
<tr>
<td>264</td>
<td>FX</td>
<td>32.088</td>
<td>MY</td>
<td>61.486</td>
</tr>
<tr>
<td>265</td>
<td>FX</td>
<td>64.176</td>
<td>MY</td>
<td>122.972</td>
</tr>
<tr>
<td>266</td>
<td>FX</td>
<td>64.176</td>
<td>MY</td>
<td>122.972</td>
</tr>
<tr>
<td>267</td>
<td>FX</td>
<td>32.088</td>
<td>MY</td>
<td>61.486</td>
</tr>
<tr>
<td>268</td>
<td>FX</td>
<td>32.088</td>
<td>MY</td>
<td>61.486</td>
</tr>
<tr>
<td>269</td>
<td>FX</td>
<td>32.088</td>
<td>MY</td>
<td>61.486</td>
</tr>
<tr>
<td>270</td>
<td>FX</td>
<td>32.088</td>
<td>MY</td>
<td>61.486</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>TOTAL =</td>
<td>577.580</td>
<td>1106.747</td>
</tr>
<tr>
<td>AT LEVEL</td>
<td>190.000 FEET</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>271</th>
<th>FX</th>
<th>35.554</th>
<th>MY</th>
<th>68.577</th>
</tr>
</thead>
<tbody>
<tr>
<td>272</td>
<td>FX</td>
<td>53.331</td>
<td>MY</td>
<td>102.866</td>
</tr>
<tr>
<td>273</td>
<td>FX</td>
<td>53.331</td>
<td>MY</td>
<td>102.866</td>
</tr>
<tr>
<td>274</td>
<td>FX</td>
<td>71.109</td>
<td>MY</td>
<td>137.154</td>
</tr>
<tr>
<td>275</td>
<td>FX</td>
<td>35.554</td>
<td>MY</td>
<td>68.577</td>
</tr>
<tr>
<td>276</td>
<td>FX</td>
<td>71.109</td>
<td>MY</td>
<td>137.154</td>
</tr>
<tr>
<td>277</td>
<td>FX</td>
<td>35.554</td>
<td>MY</td>
<td>68.577</td>
</tr>
<tr>
<td>278</td>
<td>FX</td>
<td>71.109</td>
<td>MY</td>
<td>137.154</td>
</tr>
<tr>
<td>279</td>
<td>FX</td>
<td>71.109</td>
<td>MY</td>
<td>137.154</td>
</tr>
<tr>
<td>280</td>
<td>FX</td>
<td>35.554</td>
<td>MY</td>
<td>68.577</td>
</tr>
<tr>
<td>281</td>
<td>FX</td>
<td>35.554</td>
<td>MY</td>
<td>68.577</td>
</tr>
<tr>
<td>282</td>
<td>FX</td>
<td>35.554</td>
<td>MY</td>
<td>68.577</td>
</tr>
<tr>
<td>283</td>
<td>FX</td>
<td>35.554</td>
<td>MY</td>
<td>68.577</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOTAL =</td>
<td>639.978</td>
</tr>
<tr>
<td>AT LEVEL</td>
<td>200.000 FEET</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>284</th>
<th>FX</th>
<th>39.199</th>
<th>MY</th>
<th>76.192</th>
</tr>
</thead>
<tbody>
<tr>
<td>285</td>
<td>FX</td>
<td>58.798</td>
<td>MY</td>
<td>114.288</td>
</tr>
<tr>
<td>286</td>
<td>FX</td>
<td>58.798</td>
<td>MY</td>
<td>114.288</td>
</tr>
<tr>
<td>287</td>
<td>FX</td>
<td>78.397</td>
<td>MY</td>
<td>152.384</td>
</tr>
<tr>
<td>288</td>
<td>FX</td>
<td>39.199</td>
<td>MY</td>
<td>76.192</td>
</tr>
<tr>
<td>289</td>
<td>FX</td>
<td>78.397</td>
<td>MY</td>
<td>152.384</td>
</tr>
<tr>
<td>290</td>
<td>FX</td>
<td>39.199</td>
<td>MY</td>
<td>76.192</td>
</tr>
<tr>
<td>291</td>
<td>FX</td>
<td>78.397</td>
<td>MY</td>
<td>152.384</td>
</tr>
<tr>
<td>292</td>
<td>FX</td>
<td>78.397</td>
<td>MY</td>
<td>152.384</td>
</tr>
<tr>
<td>293</td>
<td>FX</td>
<td>39.199</td>
<td>MY</td>
<td>76.192</td>
</tr>
<tr>
<td>294</td>
<td>FX</td>
<td>39.199</td>
<td>MY</td>
<td>76.192</td>
</tr>
<tr>
<td>295</td>
<td>FX</td>
<td>39.199</td>
<td>MY</td>
<td>76.192</td>
</tr>
<tr>
<td>296</td>
<td>FX</td>
<td>39.199</td>
<td>MY</td>
<td>76.192</td>
</tr>
</tbody>
</table>

| TOTAL =   | 705.575     |
| AT LEVEL  | 210.000 FEET|

<table>
<thead>
<tr>
<th>297</th>
<th>FX</th>
<th>43.021</th>
<th>MY</th>
<th>84.525</th>
</tr>
</thead>
<tbody>
<tr>
<td>298</td>
<td>FX</td>
<td>64.531</td>
<td>MY</td>
<td>126.787</td>
</tr>
<tr>
<td>299</td>
<td>FX</td>
<td>64.531</td>
<td>MY</td>
<td>126.787</td>
</tr>
<tr>
<td>300</td>
<td>FX</td>
<td>86.041</td>
<td>MY</td>
<td>169.050</td>
</tr>
<tr>
<td>301</td>
<td>FX</td>
<td>43.021</td>
<td>MY</td>
<td>84.525</td>
</tr>
<tr>
<td>302</td>
<td>FX</td>
<td>86.041</td>
<td>MY</td>
<td>169.050</td>
</tr>
<tr>
<td>303</td>
<td>FX</td>
<td>43.021</td>
<td>MY</td>
<td>84.525</td>
</tr>
<tr>
<td>304</td>
<td>FX</td>
<td>86.041</td>
<td>MY</td>
<td>169.050</td>
</tr>
<tr>
<td>305</td>
<td>FX</td>
<td>86.041</td>
<td>MY</td>
<td>169.050</td>
</tr>
<tr>
<td>306</td>
<td>FX</td>
<td>43.021</td>
<td>MY</td>
<td>84.525</td>
</tr>
<tr>
<td>307</td>
<td>FX</td>
<td>43.021</td>
<td>MY</td>
<td>84.525</td>
</tr>
<tr>
<td>308</td>
<td>FX</td>
<td>43.021</td>
<td>MY</td>
<td>84.525</td>
</tr>
<tr>
<td>309</td>
<td>FX</td>
<td>43.021</td>
<td>MY</td>
<td>84.525</td>
</tr>
</tbody>
</table>

| TOTAL =   | 774.373     |
| AT LEVEL  | 220.000 FEET|

**STAAD.Pro** 3467 **User Manual**
### CENTRE OF RIGIDITY

<table>
<thead>
<tr>
<th>DIAPHRAM</th>
<th>FL. LEVEL</th>
<th>X-COORDINATE</th>
<th>Z-COORDINATE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10.000</td>
<td>13.829</td>
<td>13.760</td>
</tr>
<tr>
<td>2</td>
<td>20.000</td>
<td>13.800</td>
<td>13.643</td>
</tr>
<tr>
<td>3</td>
<td>30.000</td>
<td>13.779</td>
<td>13.575</td>
</tr>
<tr>
<td>4</td>
<td>40.000</td>
<td>13.763</td>
<td>13.524</td>
</tr>
<tr>
<td>5</td>
<td>50.000</td>
<td>13.748</td>
<td>13.481</td>
</tr>
<tr>
<td>6</td>
<td>60.000</td>
<td>13.734</td>
<td>13.443</td>
</tr>
<tr>
<td>7</td>
<td>70.000</td>
<td>13.721</td>
<td>13.408</td>
</tr>
<tr>
<td>8</td>
<td>80.000</td>
<td>13.709</td>
<td>13.378</td>
</tr>
<tr>
<td>9</td>
<td>90.000</td>
<td>13.698</td>
<td>13.350</td>
</tr>
<tr>
<td>10</td>
<td>100.000</td>
<td>13.688</td>
<td>13.326</td>
</tr>
<tr>
<td>11</td>
<td>110.000</td>
<td>13.679</td>
<td>13.305</td>
</tr>
<tr>
<td>12</td>
<td>120.000</td>
<td>13.670</td>
<td>13.286</td>
</tr>
<tr>
<td>13</td>
<td>130.000</td>
<td>13.663</td>
<td>13.269</td>
</tr>
<tr>
<td>14</td>
<td>140.000</td>
<td>13.655</td>
<td>13.254</td>
</tr>
<tr>
<td>15</td>
<td>150.000</td>
<td>13.649</td>
<td>13.241</td>
</tr>
<tr>
<td>16</td>
<td>160.000</td>
<td>13.642</td>
<td>13.229</td>
</tr>
<tr>
<td>17</td>
<td>170.000</td>
<td>13.636</td>
<td>13.217</td>
</tr>
<tr>
<td>18</td>
<td>180.000</td>
<td>13.630</td>
<td>13.206</td>
</tr>
<tr>
<td>19</td>
<td>190.000</td>
<td>13.624</td>
<td>13.195</td>
</tr>
<tr>
<td>20</td>
<td>200.000</td>
<td>13.617</td>
<td>13.182</td>
</tr>
<tr>
<td>21</td>
<td>210.000</td>
<td>13.609</td>
<td>13.167</td>
</tr>
<tr>
<td>22</td>
<td>220.000</td>
<td>13.602</td>
<td>13.146</td>
</tr>
</tbody>
</table>

### V.IBC 2018 Static Seismic T Less Than 0.5

Verify the program-calculated base shear and its distribution along the height of a three-story frame by using the equivalent lateral force method per IBC 2018. Also, verify the torsional moments to which the floors are subjected to, considering inherent as well as accidental torsion.

**Details**

A three-story structure is subject to a seismic load from the +X direction (see following figures). The time period of the structure used is its calculated Rayleigh frequency (taken from STAAD.Pro output).
Figure 412: Plan View
Figure 413: Side Elevation (from +X dir)
**Assumptions**

i. Mapped MCER spectral response acceleration parameter at short period, $S_s = 2.02$

ii. Mapped MCER spectral response acceleration parameter at a period of 1 s, $S_1 = 0.795$

iii. Risk Category – I (Hence, From Table 1.5-2, Importance Factor $I = 1$)

iv. Site Class – D (SCLASS 4)

v. Response Modification Factor ($R_X$ & $R_Z$) = 3

vi. Long-period Transition Time ($T_L$) = 12 s

vii. Seismic weight is composed of UDLs (magnitude -5 kN/m, direction GY) (defined in Reference Load Definition), incident on the beams

viii. The effect of shear deformation is neglected

**Validation**

Calculation of Base Shear

Based on $S_s$ and $S_1$, $F_a = 1$ (per table 11.4-1) and $F_v = 1.7$ (per table 11.4-2).

Hence:

$$S_{MS} = F_a \times S_s = 1 \times 2.02 = 2.02$$

Eq. 11.4-1

$$S_{DS} = 2/3 \times S_{MS} = 1.347$$

Eq. 11.4-3
 Verification Examples
V.06 Loading

\[
S_{M1} = F_v \times S_1 = 1.7 \times 0.795 = 1.382
\]
Eq. 11.4-3

\[
S_{D1} = \frac{2}{3} \times S_{M1} = 0.901
\]
Eq. 11.4-4

\[
(C_s)_{initial} = \frac{S_{DS}}{R/T} = 0.4489
\]
Eq. 12.8-2

The time period of the structure is taken as the Rayleigh frequency of the structure (taken from the output file), \(T_R = 0.201\) s.

Height of the structure, \(h = 9\) m.

For concrete moment-resisting frames, \(C_t = 0.0466\) and \(x = 0.9\) (per Table 12.8-2).
So the approximate time period, \(T_a = C_t \times h^x = 0.0466(9\ m)^{0.9} = 0.3367\) s

From Table 12.8-1, \(C_u = 1.4\) (for \(S_{D1} > 0.4\)).

\[
C_u \times T_a = 0.4713\ s
\]

The time period used, \(T\), is the lesser of \(T_R\) and \(C_u \times T_a\), which is \(0.201\) s (< \(T_L = 12\) s).

\[
(C_s)_{max} = \frac{S_{D1}}{T \times (R/I)} = 1.4942
\]
Eq. 12.8-3

\[
(C_{S})_{min1} = 0.044 \times S_{DS} \times I = 0.0592
\]
Eq. 12.8-5

Since \(S_1 = 0.795 > 0.6g\), equation 12.8-6 also needs to be considered for calculating the lower limit of \(C_s:\)

\[
(C_s)_{min2} = \frac{0.5 \times S_1}{R/I} = 0.1325
\]
Eq. 12.8-6

Therefore, \((C_s)_{min} = 0.1325 < C_s\).

\[C_s = 0.4489\]

The seismic weight, \(W\) is taken as the total seismic weight of the beams: \(= 5 \times 3 \times 69 = 1,035\ kN\)

So the total base shear, \(V = C_s \times W = 0.4489 \times 1,035 = 464.6\ kN\)

Vertical Distribution of Lateral Forces
Since, time period of the structure \(T = 0.201\) s < 0.5 s, as per clause 12.8.3, \(k = 1\).

Hence, from equation 12.8-11 and equation 12.8-12, we can find the lateral forces in different story levels, as follows:

**Table 441: Vertical force distribution**

<table>
<thead>
<tr>
<th>Story Level</th>
<th>(W_x) (kN)</th>
<th>(h_x) (m)</th>
<th>(W_x \times h_x^k)</th>
<th>(\frac{\sum_{i=1}^{3} (W_i \times h_i^k)}{2})</th>
<th>Lateral force at story level (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>5 × 3 × 23 = 345</td>
<td>9</td>
<td>3,105</td>
<td>0.500</td>
<td>232.3</td>
</tr>
<tr>
<td>2nd</td>
<td>345</td>
<td>6</td>
<td>2,070</td>
<td>0.333</td>
<td>154.9</td>
</tr>
<tr>
<td>1st</td>
<td>345</td>
<td>3</td>
<td>1,035</td>
<td>0.167</td>
<td>77.40</td>
</tr>
<tr>
<td>(\Sigma)</td>
<td>1,035</td>
<td>-</td>
<td>6,210</td>
<td>1</td>
<td>464.6</td>
</tr>
</tbody>
</table>
Consideration of Inherent Torsion (Clause 12.8.4.1)

Floor Level – Roof (9 m)
From output file, CRZ = 3.831 m and CMZ = 3.848 m
Hence, static eccentricity esi = CMZ - CRZ = 0.017 m

Floor Level – 2nd (6 m)
From output file, CRZ = 3.858 m and CMZ = 3.848 m
Hence, static eccentricity esi = CMZ - CRZ = -0.01 m

Floor Level – 1st (3 m)
From output file, CRZ = 3.902 m and CMZ = 3.848 m
Hence, static eccentricity esi = CMZ - CRZ = -0.054 m

Consideration of Accidental Torsion (Clause 12.8.4.2)

At all floor levels:

\[ 0.05 \times L_z = 0.05 \times 9m = 0.45 \text{ m} \]

where

\[ L_z = \text{the dimension of the structure along the global Z axis} \]

Hence, total eccentricity to inherent and accidental torsion at roof level \( e_r = (0.45 + 0.017) = 0.467 \text{ m} \)

Total eccentricity to inherent and accidental torsion at 2nd floor level \( e_2 = (0.45 - 0.01) = 0.44 \text{ m} \)

Total eccentricity to inherent and accidental torsion at 1st floor level \( e_1 = (0.45 - 0.054) = 0.396 \text{ m} \)

Total torsional moment at roof level = \( F_{\text{roof}} \times e_r = 232.3 \times 0.467 = 108.48 \text{ kN·m} \)

Total torsional moment at 2nd floor level = \( F_{\text{2nd}} \times e_2 = 154.867 \times 0.44 = 68.141 \text{ kN·m} \)

Total torsional moment at 1st floor level = \( F_{\text{1st}} \times e_1 = 77.433 \times 0.396 = 30.664 \text{ kN·m} \)

Results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base shear, V (kN)</td>
<td>464.6</td>
<td>464.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Lateral force at roof level (kN)</td>
<td>232.3</td>
<td>232.3</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Lateral force at 2nd floor (kN)</td>
<td>154.9</td>
<td>154.867</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Lateral force at 1st floor (kN)</td>
<td>77.4</td>
<td>77.433</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Torsional moment at roof level (kN-m)</td>
<td>108.48</td>
<td>108.523</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>
### Result Type

<table>
<thead>
<tr>
<th></th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Torsional moment at 2nd floor (kN·m)</td>
<td>68.141</td>
<td>68.092</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Torsional moment at 1st floor (kN·m)</td>
<td>30.664</td>
<td>30.619</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\2018\IBC 2018 Static Seismic T Less Than 0.5.STD is typically installed with the program.

**STAAD SPACE**

**START JOB INFORMATION**

**ENGINEER DATE** 08-Mar-19

**END JOB INFORMATION**

*This problem is created to verify the base shear, distribution of base shear *And Inherent and Accidental Torsional Moment at different floor levels *Of the Structure

**INPUT WIDTH 79**

**SET SHEAR**

**UNIT METER KN**

**JOINT COORDINATES**

| 1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 0 3; 6 0 3 3; 7 3 3 3; 8 3 0 3; 9 0 0 6; 10 0 3 6; 11 3 3 6; 12 3 0 6; 13 0 0 9; 14 0 3 9; 15 3 3 9; 16 3 0 9; 17 6 3 0; 18 6 0 0; 19 6 3 3; 20 6 0 3; 21 6 3 6; 22 6 0 6; 25 9 3 3; 26 9 0 3; 27 9 3 6; 28 9 0 6; 30 6 0 0; 35 0 6 0; 36 3 6 3; 37 0 6 6; 38 3 6 6; 39 0 6 9; 40 3 6 9; 41 6 6 0; 42 6 6 3; 43 6 6 6; 45 9 6 3; 46 9 6 6; 49 0 9 0; 50 3 9 0; 51 0 9 3; 52 3 9 3; 53 0 9 6; 54 3 9 6; 55 0 9 9; 56 3 9 9; 57 6 9 0; 58 6 9 3; 59 6 9 6; 70 9 9 3; 71 9 9 6; 72 -3 0 0; 73 -3 3 0; 74 -3 0 3; 75 -3 3 3; 76 -3 0 6; 77 -3 3 6; 80 -3 6 0; 81 -3 6 3; 82 -3 6 6; 84 -3 9 0; 85 -3 9 3; 86 -3 9 6;

**MEMBER INCIDENCES**

| 1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11; 11 9 10; 12 10 11; 13 11 12; 14 10 14; 15 11 15; 16 13 14; 17 14 15; 18 15 16; 19 3 17; 20 7 19; 21 11 21; 23 17 18; 24 17 19; 25 19 20; 26 19 21; 27 21 22; 30 19 25; 31 21 27; 32 25 26; 33 25 27; 34 27 28; 40 2 33; 41 3 34; 42 6 35; 43 7 36; 44 10 37; 45 11 38; 46 14 39; 47 15 40; 48 17 41; 49 19 42; 50 21 43; 52 25 45; 53 27 46; 56 33 34; 57 33 35; 58 34 36; 59 35 36; 60 35 37; 61 36 38; 62 37 38; 63 37 39; 64 38 40; 65 39 40; 66 34 41; 67 36 42; 68 38 43; 70 41 42; 71 42 43; 73 42 45; 74 43 46; 75 45 46; 79 33 49; 80 34 50; 81 35 51; 82 36 52; 83 37 53; 84 38 54; 85 39 55; 86 40 56; 87 41 57; 88 42 58; 89 43 59; 95 49 50; 96 49 51; 97 50 52; 98 51 53; 99 51 53; 100 52 54; 101 53 54; 102 53 55; 103 54 56; 104 55 56; 105 50 57; 106 52 58; 107 54 59; 109 57 58; 110 58 59;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL

MEMBER PROPERTY AMERICAN
1 TO 21 23 TO 34 40 TO 50 52 53 56 TO 68 70 71 73 TO 75 79 TO 89 -
95 TO 107 109 110 128 129 132 TO 137 139 TO 141 143 TO 145 147 TO 151 154 -
155 TO 156 158 159 161 TO 163 165 166 PRIS YD 0.4 ZD 0.4

CONSTANTS
MATERIAL CONCRETE ALL

SUPPORTS
1 4 5 8 9 12 13 16 18 20 22 26 28 72 74 76 FIXED

DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1

MEMBER LOAD
2 4 5 7 9 10 12 14 15 17 19 TO 21 24 26 30 31 33 56 TO 68 70 71 73 TO 75 95 -
96 TO 107 109 110 132 TO 137 139 TO 141 143 TO 145 147 TO 151 154 -
155 TO 166 UNI GY -5

END DEFINE REFERENCE LOADS

FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9

DEFINE IBC 2018
SS 2.02 S1 0.795 I 1 RX 3 RZ 3 SCLASS 4 TL 12

LOAD 1 LOADTYPE Seismic  TITLE SL +X
IBC LOAD X 1 DEC 2 ACC 0.05

PERFORM ANALYSIS PRINT LOAD DATA
PRINT DIA CR

FINISH

---

**STAAD.Pro Output**

****************************************************************************************************
* EQUIV. SEISMIC LOADS AS PER IBC

* PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD GENERATION

* SS = 2.020 S1 = 0.795 FA = 1.000 FV = 1.700
* SDS = 1.347 SD1 = 0.901

****************************************************************************************************

72. LOAD 1 LOADTYPE SEISMIC  TITLE SL +X

---

STAAD.Pro 3475 User Manual
73. IBC LOAD X 1 DEC 2 ACC 0.05
74. PERFORM ANALYSIS PRINT LOAD DATA

PROBLEM STATISTICS
----------------------------------------------------------
NUMBER OF JOINTS         67  NUMBER OF MEMBERS     117
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     162
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

LOADING  1  LOADTYPE SEISMIC TITLE SL +X
----------

******************************************************************************
***NOTE: SEISMIC LOAD IS ACTING AT CENTER OF MASS FOR RIGID DIAPHRAGM.
TORISION FROM STATIC ECCENTRICITY (esi) IS INCLUDED IN ANALYSIS.
DYNAMIC ECCENTRICITY APPLIED = DEC - 1

LOAD NO.:     1  DIRECTION : X   UNIT - METE
STORY LEVEL     DYN. ECC. (dec)     ACC. ECC. (aec)     DESIGN     ECC.
-----  -----     ---------------     ---------------     ---------------
        X         Z         X         Z         X         Z
        dec + aec dec + aec
1     3.00      -0.05     -0.05      0.60      0.45      0.00      0.40
2     6.00       0.01     -0.01      0.60      0.45      0.00      0.44
3     9.00      -0.05      0.02      0.60      0.45      0.00      0.47

******************************************************************************
JOINT LATERAL TORSIONAL LOAD - 1
LOAD (KN )  MOMENT (KN -Mete) FACTOR - 1.000
-----  -----------------------------------------------
        DEC + AEC
1    2    FX    5.050    MY    1.997
3    3    FX    5.050    MY    1.997
6    6    FX    6.733    MY    2.663
7    7    FX    6.733    MY    2.663
10   10   FX    6.733    MY    2.663
11   11   FX    6.733    MY    2.663
14   14   FX    3.367    MY    1.331
15   15   FX    3.367    MY    1.331
17   17   FX    3.367    MY    1.331
19   19   FX    6.733    MY    2.663
21   21   FX    5.050    MY    1.997
25   25   FX    3.367    MY    1.331
27   27   FX    3.367    MY    1.331

Verification Examples

STAAD.Pro

3476

User Manual
Verification Examples
V.06 Loading

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>73</td>
<td></td>
<td></td>
<td>1.331</td>
</tr>
<tr>
<td>75</td>
<td></td>
<td></td>
<td>1.997</td>
</tr>
<tr>
<td>77</td>
<td></td>
<td></td>
<td>1.331</td>
</tr>
</tbody>
</table>

TOTAL = 77.433 AT LEVEL 3.000 METE

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>33</td>
<td>10.100</td>
<td></td>
<td>4.441</td>
</tr>
<tr>
<td>34</td>
<td>10.100</td>
<td></td>
<td>4.441</td>
</tr>
<tr>
<td>35</td>
<td>13.467</td>
<td></td>
<td>5.921</td>
</tr>
<tr>
<td>36</td>
<td>13.467</td>
<td></td>
<td>5.921</td>
</tr>
<tr>
<td>37</td>
<td>13.467</td>
<td></td>
<td>5.921</td>
</tr>
<tr>
<td>38</td>
<td>13.467</td>
<td></td>
<td>5.921</td>
</tr>
<tr>
<td>39</td>
<td>6.733</td>
<td></td>
<td>2.961</td>
</tr>
<tr>
<td>40</td>
<td>6.733</td>
<td></td>
<td>2.961</td>
</tr>
<tr>
<td>41</td>
<td>6.733</td>
<td></td>
<td>2.961</td>
</tr>
<tr>
<td>42</td>
<td>13.467</td>
<td></td>
<td>5.921</td>
</tr>
<tr>
<td>43</td>
<td>10.100</td>
<td></td>
<td>4.441</td>
</tr>
<tr>
<td>45</td>
<td>6.733</td>
<td></td>
<td>2.961</td>
</tr>
<tr>
<td>46</td>
<td>6.733</td>
<td></td>
<td>2.961</td>
</tr>
<tr>
<td>80</td>
<td>6.733</td>
<td></td>
<td>2.961</td>
</tr>
<tr>
<td>81</td>
<td>10.100</td>
<td></td>
<td>4.441</td>
</tr>
<tr>
<td>82</td>
<td>6.733</td>
<td></td>
<td>2.961</td>
</tr>
</tbody>
</table>

TOTAL = 154.867 AT LEVEL 6.000 METE

<table>
<thead>
<tr>
<th></th>
<th>FX</th>
<th>MY</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>49</td>
<td>15.150</td>
<td></td>
<td>7.078</td>
</tr>
<tr>
<td>50</td>
<td>15.150</td>
<td></td>
<td>7.078</td>
</tr>
<tr>
<td>51</td>
<td>20.200</td>
<td></td>
<td>9.437</td>
</tr>
<tr>
<td>52</td>
<td>20.200</td>
<td></td>
<td>9.437</td>
</tr>
<tr>
<td>53</td>
<td>20.200</td>
<td></td>
<td>9.437</td>
</tr>
<tr>
<td>54</td>
<td>20.200</td>
<td></td>
<td>9.437</td>
</tr>
<tr>
<td>55</td>
<td>10.100</td>
<td></td>
<td>4.718</td>
</tr>
<tr>
<td>56</td>
<td>10.100</td>
<td></td>
<td>4.718</td>
</tr>
<tr>
<td>57</td>
<td>10.100</td>
<td></td>
<td>4.718</td>
</tr>
<tr>
<td>58</td>
<td>20.200</td>
<td></td>
<td>9.437</td>
</tr>
<tr>
<td>59</td>
<td>15.150</td>
<td></td>
<td>7.078</td>
</tr>
<tr>
<td>70</td>
<td>10.100</td>
<td></td>
<td>4.718</td>
</tr>
<tr>
<td>71</td>
<td>10.100</td>
<td></td>
<td>4.718</td>
</tr>
<tr>
<td>84</td>
<td>10.100</td>
<td></td>
<td>4.718</td>
</tr>
<tr>
<td>85</td>
<td>15.150</td>
<td></td>
<td>7.078</td>
</tr>
<tr>
<td>86</td>
<td>10.100</td>
<td></td>
<td>4.718</td>
</tr>
</tbody>
</table>

STAAD SPACE -- PAGE NO.

TOTAL = 232.300 AT LEVEL 9.000 METE

*********** END OF DATA FROM INTERNAL STORAGE ***********

75. PRINT DIA CR
DIA CR

CENTRE OF RIGIDITY UNIT - METE

<table>
<thead>
<tr>
<th></th>
<th>FL. LEVEL</th>
<th>X-COORDINATE</th>
<th>Z-COORDINATE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.000</td>
<td>2.393</td>
<td>3.902</td>
</tr>
<tr>
<td>2</td>
<td>6.000</td>
<td>2.335</td>
<td>3.858</td>
</tr>
<tr>
<td>3</td>
<td>9.000</td>
<td>2.299</td>
<td>3.831</td>
</tr>
</tbody>
</table>

---

Related Links
- TR.31.2.15 IBC 2018 Seismic Load Definition (on page 2603)
V. IS 1893

V. IS 1893 2002

V. IS 1893 2002 Response Spectrum


Reference

1. Example 4.7, http://nptel.ac.in/courses/105101004/4 (p.36)

Related Links

• TR.32.10.1.6 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2721)

Problem

The reference example given in (1) is modelled in STAAD.Pro considering the following:

- Seismic zone factor, Z = 0.36
- Importance factor, I = 1
- Response reduction factor, R = 5
- Soil site conditions = Hard

The generated model is subjected to a response spectrum load along global X direction. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 443: Natural Frequencies/Time Period

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (cyc/sec)</th>
<th>Time Period (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

Table 444: Mode Shapes

<table>
<thead>
<tr>
<th>Mode</th>
<th>Story Level/Joint Numbers</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
## Mode Story Level/Joint Numbers

<table>
<thead>
<tr>
<th>Mode</th>
<th>Story Level/Joint Numbers</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>2nd Floor/Joints 5 &amp; 6</td>
<td>0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/Joints 5 &amp; 6</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/Joints 2 &amp; 3</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/Joints 5 &amp; 6</td>
<td>-0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### Table 445: Mode Participation Factor Calculation

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$\phi$</td>
<td>$Wi \times \phi$</td>
<td>$Wi \times \phi^2$</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>49.035</td>
<td>49.035</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86603</td>
<td>84.932</td>
<td>73.553</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>49.035</td>
<td>24.518</td>
</tr>
<tr>
<td>Summation</td>
<td>245.175</td>
<td>183.00</td>
<td>147.11</td>
<td>147.105</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Modal Weight $Mk \times g$</th>
<th>227.66</th>
<th>16.345</th>
<th>1.173</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modal Weight Participation (In %)</td>
<td>92.85</td>
<td>6.667</td>
<td>0.479</td>
</tr>
<tr>
<td>Mode Participation Factor $P_k \sum Wi \times \phi / \sum Wi \times \phi^2$</td>
<td>1.244</td>
<td>0.3333</td>
<td>0.0893</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$S_o/g$</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2.5 (time period &lt; 0.4 s)</td>
<td>2.5 (time period &lt; 0.4 s)</td>
<td>2.2063 (1 + 15 \times 0.08042 as time period &lt; 0.1 s)</td>
</tr>
</tbody>
</table>
Mode 1 Mode 2 Mode 3

\( Ak \left( \frac{Z}{2} \times I/R \times S_a/g \right) \)

\[ 0.09 \] \[ (0.36/2 \times 1/5 \times 2.5) \]
\[ 0.09 \] \[ (0.36/2 \times 1/5 \times 2.5) \]
\[ 0.0794 \] \[ (0.36/2 \times 1/5 \times 2.2063) \]

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Story Shear due to all modes i.e. SRSS of all modes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>( \phi )</td>
<td>Qi (( Ak \times \phi \times Pk \times Wi ))</td>
<td>Vi (( \sum ) Qi)</td>
<td>( \phi )</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>5.490</td>
<td>5.490</td>
<td>-1</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.8660</td>
<td>9.509</td>
<td>15.0</td>
<td>0</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>5.490</td>
<td>20.49</td>
<td>1</td>
</tr>
</tbody>
</table>

Base Shear 20.54

Comparison

Table 446: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Roof</td>
<td>5.69</td>
<td>5.69</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>2nd</td>
<td>15.07</td>
<td>15.07</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>1st</td>
<td>20.54</td>
<td>20.54</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Base shear (kN)</td>
<td>20.54</td>
<td>20.54</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2002\IS 1893 2002 Response Spectrum.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLumn
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLumn
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS 1893 X 0.036 ACC DAMP 0.05
SOIL TYPE 1
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH

STAAD Output
1893 RESPONSE SPECTRUM LOAD 1
RESPONSE LOAD CASE 1
**Verification Examples**

**V.06 Loading**

---

### Modal Weight (Modal Mass Times g) in KN

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.276565E+02</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>2</td>
<td>1.634500E+01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>3</td>
<td>1.173519E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471051E+02</td>
</tr>
</tbody>
</table>

**SRSS Modal Combination Method Used.**

**Dynamic Weight X Y Z** | 2.451750E+02 | 0.000000E+00 | 0.000000E+00 KN

**Missing Weight X Y Z** | -3.177132E-06 | 0.000000E+00 | 0.000000E+00 KN

**Modal Weight X Y Z** | 2.451750E+02 | 0.000000E+00 | 0.000000E+00 KN

---

### Response Load Case 1

**Mode** | **Spectral Acceleration** | **Design Seismic Coefficient**

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.50000</td>
<td>0.0900</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>2.50000</td>
<td>0.0900</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>2.20632</td>
<td>0.0794</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

---

### Peak Story Shear

<table>
<thead>
<tr>
<th>Story</th>
<th>Level in Metre</th>
<th>Peak Story Shear in KN</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
<td>Z</td>
</tr>
<tr>
<td>3</td>
<td>9.00</td>
<td>5.69</td>
</tr>
<tr>
<td>2</td>
<td>6.00</td>
<td>15.07</td>
</tr>
<tr>
<td>1</td>
<td>3.00</td>
<td>20.54</td>
</tr>
<tr>
<td>Base</td>
<td>0.00</td>
<td>20.54</td>
</tr>
</tbody>
</table>

---

### Response Spectrum Values - Units (Metre Second)

**Directional Values:**

- Scale Factor = 1.00
- Damping Factor = 0.050

**Period vs. Acceleration**

<table>
<thead>
<tr>
<th>Period</th>
<th>Acceleration-G</th>
<th>Damping</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.300</td>
<td>24.5166</td>
<td></td>
</tr>
<tr>
<td>0.1099</td>
<td>24.5166</td>
<td></td>
</tr>
<tr>
<td>0.0804</td>
<td>21.6366</td>
<td></td>
</tr>
</tbody>
</table>

---

### Modal Base Actions

**Modal Base Actions**

<table>
<thead>
<tr>
<th>Forces in KN</th>
<th>Length in Metre</th>
</tr>
</thead>
</table>

**Moments Are**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Period</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.300</td>
<td>20.49</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>0.00</td>
<td>-122.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.110</td>
<td>1.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>0.00</td>
<td>4.41</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.080</td>
<td>0.09</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>0.00</td>
<td>-0.56</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

### Participation Factors

**Mass Participation Factors in Percent**

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Summ-X</th>
<th>Summ-Y</th>
<th>Summ-Z</th>
</tr>
</thead>
</table>

---

**STAAD SPACE** -- PAGE NO.

---

**STAAD Pro** 3482 User Manual
V. IS 1893 2002 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static method in IS 1893 (Part 1) : 2002

Problem

A reinforced concrete frame structure is 4 bays at 4m ea. by 3 bays at 4m ea. in plan. The structure is 3 stories at 3m ea.

The structure is modelled in STAAD.Pro considering the following:

- Seismic zone factor, \( Z = 0.36 \) (ZONE 0.36)
- Importance factor, \( I = 1 \) (I 1)
- Response reduction factor, \( R = 5 \) (RF 5)
- Soil site condition = Hard (SS 1)
- Structure type is a reinforced concrete frame (ST 1)

The seismic weight for each floor is assumed as 100 kN applied as joint weight to nodes 39, 60, & 80. Seismic Loads as per IS 1893 (Part 1) : 2002 specifications are generated along two horizontal directions global X & global Z. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Height = 3 \( \times \) 3 = 9m

\[
T_a_x = 0.075h^{0.75} = 0.075 \times (9^{0.75}) = 0.39 \text{ sec}
\]

\[
T_a_z = 0.075h^{0.75} = 0.075 \times (9^{0.75}) = 0.39 \text{ sec}
\]

\[
[Sa/g]_x = [Sa/g]_z = 2.5 \text{ (As time period < 0.4 sec)}
\]

\[
A_h_x = (Z/2) \times (I/R) \times [Sa/g]_x = 0.36/2 \times 1/5 \times 2.5 = 0.09
\]

\[
A_h_z = (Z/2) \times (I/R) \times [Sa/g]_z = 0.36/2 \times 1/5 \times 2.5 = 0.09
\]

\[
V_B = A_h \times W
\]

\[
V_{Bx} = A_h_x \times W = 0.09 \times (3 \times 100) = 27 \text{ kN}
\]

\[
V_{Bz} = A_h_z \times W = 0.09 \times (3 \times 100) = 27 \text{ kN}
\]
### Story Level Summary

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Wi</th>
<th>hi</th>
<th>$Wi \times hi^2$</th>
<th>$(Wi \times hi^2)/\sum(Wi \times hi^2)$</th>
<th>Qx</th>
<th>Qz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>100</td>
<td>9</td>
<td>8,100</td>
<td>0.643</td>
<td>17.357</td>
<td>17.357</td>
</tr>
<tr>
<td>2nd</td>
<td>100</td>
<td>6</td>
<td>3,600</td>
<td>0.286</td>
<td>7.714</td>
<td>7.714</td>
</tr>
<tr>
<td>1st</td>
<td>100</td>
<td>3</td>
<td>900</td>
<td>0.0714</td>
<td>1.929</td>
<td>1.929</td>
</tr>
<tr>
<td>Summation</td>
<td>300</td>
<td></td>
<td>12,600</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Vb_h

<table>
<thead>
<tr>
<th>Vb_h</th>
<th>27</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Comparison

**Table 447: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear X, Z (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Roof</td>
<td>1.929, 1.929</td>
<td>1.929, 1.929</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>2nd</td>
<td>7.714, 7.714</td>
<td>7.714, 7.714</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>1st</td>
<td>17.357, 17.357</td>
<td>17.357, 17.357</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Base Shear (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>VBx</td>
<td>27</td>
<td>27.000</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>VBz</td>
<td>27</td>
<td>27.000</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2002\IS 1893 2002 Static Seismic.STD` typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Jun-18
END JOB INFORMATION
INPUT WIDTH 79
*SET STAR 0
UNIT METER KN
JOINT COORDINATES
  1 0 0 0; 2 0 3 0; 3 4 3 0; 4 4 0 0; 5 0 0 4; 6 0 3 4; 7 4 3 4; 8 4 0 4;
  9 8 3 0; 10 8 0 0; 11 8 3 4; 12 8 0 4; 13 12 3 0; 14 12 0 0; 15 12 3 4;
  16 12 0 4; 17 16 3 0; 18 16 0 0; 19 16 3 4; 20 16 0 4; 21 0 0 8; 22 0 3 8;
  23 4 3 8; 24 4 0 8; 25 8 3 8; 26 8 0 8; 27 12 3 8; 28 12 0 8; 29 16 3 8;
  30 16 0 8; 31 0 0 12; 32 0 3 12; 33 4 3 12; 34 4 0 12; 35 8 3 12; 36 8 0 12;
  37 12 3 12; 38 12 0 12; 39 16 3 12; 40 16 0 12; 41 0 6 0; 42 4 6 0; 43 0 6 4;
  44 4 6 4; 45 8 6 0; 46 8 6 4; 47 12 6 0; 48 12 6 4; 49 16 6 0; 50 16 6 4;
  51 0 6 8; 52 4 6 8; 53 8 6 8; 54 12 6 8; 55 16 6 8; 56 0 6 12; 57 4 6 12;
  58 8 6 12; 59 12 6 12; 60 16 6 12; 61 0 9 0; 62 4 9 0; 63 0 9 4; 64 4 9 4;
  65 8 9 0; 66 8 9 4; 67 12 9 0; 68 12 9 4; 69 16 9 0; 70 16 9 4;
  71 0 9 8; 72 4 9 8; 73 8 9 8; 74 12 9 8; 75 16 9 8; 76 0 9 12;
```
Verification Examples

V.06 Loading

77 4 9 12; 78 8 9 12; 79 12 9 12; 80 16 9 12;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 3 9; 10 7 11;
11 9 10; 12 9 11; 13 11 12; 14 9 13; 15 11 15; 16 13 14; 17 13 15; 18 15 16;
19 13 17; 20 15 19; 21 17 18; 22 17 19; 23 19 20; 24 6 22; 25 7 23; 26 11 25;
27 15 27; 28 19 29; 29 21 22; 30 22 23; 31 23 24; 32 23 25; 33 25 26; 34 25 27;
35 27 28; 36 27 29; 37 29 30; 38 22 32; 39 23 33; 40 25 35; 41 27 37; 42 29 39;
43 31 32; 44 32 33; 45 33 34; 46 33 35; 47 35 36; 48 35 37; 49 37 38; 50 37 39;
51 39 40; 52 2 41; 53 3 42; 54 6 43; 55 7 44; 56 9 45; 57 11 46; 58 13 47;
59 15 48; 60 17 49; 61 19 50; 62 22 51; 63 23 52; 64 25 53; 65 27 54; 66 29 55;
67 32 56; 68 33 57; 69 35 58; 70 37 59; 71 39 60; 72 41 42; 73 41 43; 74 42 44;
75 43 44; 76 42 45; 77 44 46; 78 45 46; 79 45 47; 80 46 48; 81 47 48; 82 47 49;
83 48 50; 84 49 50; 85 43 51; 86 44 52; 87 46 53; 88 48 54; 89 50 55; 90 51 52;
91 52 53; 92 53 54; 93 54 55; 94 51 56; 95 52 57; 96 53 58; 97 54 59; 98 55 60;
99 56 57; 100 57 58; 101 58 59; 102 59 60; 103 41 61; 104 42 62; 105 43 63;
106 44 64; 107 45 65; 108 46 66; 109 47 67; 110 48 68; 111 49 69; 112 50 70;
113 51 71; 114 52 72; 115 53 73; 116 54 74; 117 55 75; 118 56 76; 119 57 77;
120 58 78; 121 59 79; 122 60 80; 123 61 62; 124 61 63; 125 62 64; 126 63 64;
127 62 65; 128 64 66; 129 65 66; 130 65 67; 131 66 68; 132 67 68; 133 67 69;
134 68 70; 135 69 70; 136 63 71; 137 64 72; 138 66 73; 139 68 74; 140 70 75;
141 71 72; 142 72 73; 143 73 74; 144 74 75; 145 71 76; 146 72 77; 147 73 78;
148 74 79; 149 75 80; 150 76 77; 151 77 78; 152 78 79; 153 79 80;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 153 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 10 12 14 16 18 20 21 24 26 28 30 31 34 36 38 40 FIXED
DEFINE 1893 LOAD
*DEFINE IS1893 2016 LOAD
*ZONE 0.36 RF 5 I 1.2 SS 1 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1 SS 1 ST 1 DM 0.05
*ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05
*ZONE 0.36 RF 5 I 1 SS 3 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1 SS 3 ST 1 DM 0.05
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
*LOAD 3 LOADTYPE Seismic  TITLE SS_(+Y)
*1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH

STAAD Output

*****************************************************************************
* UNITS - KN    METE                                    *
* TIME PERIOD FOR X 1893 LOADING =    0.38971 SEC       *
* SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
* VB PER 1893=    0.0900 X      300.00=      27.00 KN   *
*                                                       *
*****************************************************************************

*****************************************************************************
* UNITS - KN    METE                                    *
* TIME PERIOD FOR Z 1893 LOADING =    0.38971 SEC       *
* SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
* VB PER 1893=    0.0900 X      300.00=      27.00 KN   *
*                                                       *
*****************************************************************************

Joint            Lateral          Torsional            Load - 1
                Moment (KN -METE) Factor - 1.000
-------          -------          --------
39              FX                1.929    MY          0.000

TOTAL = 1.929 0.000 AT LEVEL 3.000 METE
VB PER 1893 = 27.000 KN

60              FX                7.714    MY          0.000

TOTAL = 7.714 0.000 AT LEVEL 6.000 METE
VB PER 1893 = 27.000 KN

80              FX                17.357   MY         0.000

TOTAL = 17.357 0.000 AT LEVEL 9.000 METE
VB PER 1893 = 27.000 KN

STAAD SPACE

Joint            Lateral          Torsional            Load - 2
                Moment (KN -METE) Factor - 1.000
-------          -------          --------
39              FZ                1.929    MY          0.000

TOTAL = 1.929 0.000 AT LEVEL 3.000 METE
VB PER 1893 = 27.000 KN

60              FZ                7.714    MY          0.000

TOTAL = 7.714 0.000 AT LEVEL 6.000 METE
VB PER 1893 = 27.000 KN

80              FZ                17.357   MY         0.000

TOTAL = 17.357 0.000 AT LEVEL 9.000 METE
VB PER 1893 = 27.000 KN
Verification Examples
V.06 Loading

Related Links

V. IS 1893 2016

V. IS 1893 2016 Response Spectrum

Reference
1. Example 4.7, http://nptel.ac.in/courses/105101004/4 (p.36)

Related Links
• TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2016 (on page 2732)

Problem
The reference example given in (1) is modelled in STAAD.Pro considering the following:

Seismic zone factor, Z = 0.36
Importance factor, I = 1.2
Response reduction factor, R = 5
Soil site conditions = Hard

The generated model is subjected to a response spectrum load along global X direction. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 448: Natural Frequencies/Time Period

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (cyc/sec)</th>
<th>Time Period (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

Table 449: Mode Shapes

<table>
<thead>
<tr>
<th>Mode</th>
<th>Story Level/Joint Numbers</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/Joints 5 &amp; 6</td>
<td>0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
### Mode Story Level/Joint Numbers

<table>
<thead>
<tr>
<th>Mode</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>-0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### Table 450: Mode Participation Factor Calculation

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>(\phi)</td>
<td>(Wi \times \phi)</td>
<td>(Wi \times \phi^2)</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>49.035</td>
<td>49.035</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86</td>
<td>84.932</td>
<td>73.553</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>49.035</td>
<td>24.518</td>
</tr>
<tr>
<td>Summation</td>
<td>245.175</td>
<td>183.00</td>
<td>147.11</td>
<td>147.105</td>
</tr>
<tr>
<td>Modal Weight (M_k \times g)</td>
<td>227.66</td>
<td>16.345</td>
<td>1.173</td>
<td></td>
</tr>
<tr>
<td>Modal Weight Participation (In %)</td>
<td>92.85</td>
<td>6.667</td>
<td>0.479</td>
<td></td>
</tr>
<tr>
<td>Modal Participation Factor (P_k \sum Wi \times \phi / \sum Wi \times \phi^2)</td>
<td>1.244</td>
<td>0.3333</td>
<td>0.0893</td>
<td></td>
</tr>
</tbody>
</table>

### Modal Response

<table>
<thead>
<tr>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>(S_u/g) = 2.5 (time period &lt; 0.4 s)</td>
<td>2.5 (time period &lt; 0.4 s)</td>
<td>2.2063 (1 + 15 \times 0.08042 as time period &lt; 0.1 s)</td>
</tr>
</tbody>
</table>
### Mode 1

| Ak (Z/2 × I/R × Sa/g) | 0.108 (0.36/2 × 1.2/5 × 2.5) | 0.108 (0.36/2 × 1.2/5 × 2.5) | 0.09531216 (0.36/2 × 1.2/5 × 2.2063) |

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Story Shear due to all modes i.e. SRSS of all modes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>6.588</td>
<td>6.588</td>
<td>-1</td>
<td>1.765</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.8660</td>
<td>11.411</td>
<td>18.0</td>
<td>0</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>6.588</td>
<td>24.587</td>
<td>1</td>
</tr>
</tbody>
</table>

### Comparison

**Table 451: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear (kN)</td>
<td>Roof</td>
<td>6.83</td>
<td>6.83</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td>2nd</td>
<td>18.09</td>
<td>18.09</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td>1st</td>
<td>24.65</td>
<td>24.65</td>
<td>none</td>
</tr>
<tr>
<td>Base shear (kN)</td>
<td></td>
<td>24.65</td>
<td>24.65</td>
<td>none</td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2016\IS 1893 2016 Response Spectrum.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
```
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLumn
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLumn
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS IS1893 2016 X 0.0432 DAMP 0.05
SOIL TYPE 1
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>RESPONSE SPECTRUM LOAD</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>RESPONSE LOAD CASE</td>
<td>1</td>
</tr>
</tbody>
</table>
### Modal Weight (Modal Mass Times g) in KN

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.276565E+02</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>2</td>
<td>1.634500E+01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>3</td>
<td>1.173519E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471051E+02</td>
</tr>
</tbody>
</table>

**SRSS Modal Combination Method Used.**

**Dynamic Weight X Y Z** 2.451750E+02
**Missing Weight X Y Z** -3.177132E-06
**Modal Weight X Y Z** 2.451750E+02

### Response Load Case 1

<table>
<thead>
<tr>
<th>Mode</th>
<th>Spectral Acceleration</th>
<th>Design Seismic Coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.50000</td>
<td>0.1080</td>
</tr>
<tr>
<td>2</td>
<td>2.50000</td>
<td>0.1080</td>
</tr>
<tr>
<td>3</td>
<td>2.20632</td>
<td>0.0953</td>
</tr>
</tbody>
</table>

### Staad Space

#### Peak Story Shear

<table>
<thead>
<tr>
<th>Story</th>
<th>Level in Mete</th>
<th>Peak Story Shear in KN</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>9.00</td>
<td>6.83</td>
</tr>
<tr>
<td>2</td>
<td>6.00</td>
<td>18.09</td>
</tr>
<tr>
<td>1</td>
<td>3.00</td>
<td>24.65</td>
</tr>
<tr>
<td>Base</td>
<td>0.00</td>
<td>24.65</td>
</tr>
</tbody>
</table>

### Response Spectrum Values - Units (Mete Second)

- **Directional Values:**
  - Scale Factor = 1.00
  - Damping Factor = 0.050

### Modal Base Actions

<table>
<thead>
<tr>
<th>Mode</th>
<th>Period</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.300</td>
<td>24.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.110</td>
<td>1.77</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>
V. IS 1893 2016 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static method in IS 1893 (Part 1): 2016

**Problem**

A reinforced concrete frame structure is 4 bays at 4m ea. by 3 bays at 4m ea. in plan. The structure is 3 stories at 3m ea.

The structure is modelled in STAAD.Pro considering the following:

- Seismic zone factor, $Z = 0.36$ (ZONE 0.36)
- Importance factor, $I = 1.2$ (I 1.2)
- Response reduction factor, $R = 5$ (RF 5)
- Soil site condition = Hard (SS 1)
- Structure type is a reinforced concrete frame (ST 1)

The seismic weight for each floor is assumed as 100 kN applied as joint weight to nodes 39, 60 & 80. Seismic Loads as per IS 1893 (Part 1): 2016 specifications are generated along two horizontal directions global X & global Z and also along vertical direction global Y. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Height = 3 × 3 = 9m

$$T_a = 0.075h^{0.75} = 0.075 \times (9^{0.75}) = 0.39 \text{ sec}$$

$$[Sa/g]_x = [Sa/g]_y = 2.5 \text{ (As time period < 0.4 sec)}$$

$$[Sa/g]_y = 2.5 \text{ (constant)}$$
Ah_x = (Z/2) × (I/R) × [Sa/g]_x = 0.36/2 × 1.2/5 × 2.5 = 0.108
Ah_z = (Z/2) × (I/R) × [Sa/g]_z = 0.36/2 × 1.2/5 × 2.5 = 0.108
Av = (2/3) × (Z/2) × (I/R) × [Sa/g]_y = 2/3 × 0.36/2 × 1.2/5 × 2.5 = 0.072
VB = Ah × W or Av × W
VB_x = Ah_x × W = 0.108 × (3 × 100) = 32.4 kN
VBmin_x = 2.4% of W = 2.4 × (3 × 100) / 100 = 7.2 kN
VB_x > VBmin_x
VB_z = Ah_z × W = 0.108 × (3 × 100) = 32.4 kN
VBmin_z = 2.4% of W = 2.4 × (3 × 100) / 100 = 7.2 kN
VB_z > VBmin_z
VB_y = Av × W = 0.072 × (3 × 100) = 21.6 kN

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Wi</th>
<th>hi</th>
<th>Wi × hi²</th>
<th>(Wi × hi²) / Σ(Wi × hi²)</th>
<th>Qx</th>
<th>Qz</th>
<th>Qy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>100</td>
<td>9</td>
<td>8,100</td>
<td>0.6429</td>
<td>20.83</td>
<td>20.83</td>
<td>13.886</td>
</tr>
<tr>
<td>2nd</td>
<td>100</td>
<td>6</td>
<td>3,600</td>
<td>0.2857</td>
<td>9.257</td>
<td>9.257</td>
<td>6.171</td>
</tr>
<tr>
<td>1st</td>
<td>100</td>
<td>3</td>
<td>900</td>
<td>0.0714</td>
<td>2.314</td>
<td>2.314</td>
<td>1.543</td>
</tr>
<tr>
<td>Summation</td>
<td>300</td>
<td></td>
<td>12,600</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Vb_h = 32.4

Vbv = 21.6

Vb_h(min) = 7.2

Comparison

Table 452: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear X, Z, Y (kN)</td>
<td>Roof</td>
<td>2.314, 2.314, 1.543</td>
<td>2.314, 2.314, 1.543</td>
<td>none</td>
</tr>
<tr>
<td>Base shear (kN)</td>
<td>VB_x</td>
<td>32.4</td>
<td>32.4</td>
<td>none</td>
</tr>
<tr>
<td>Parameter</td>
<td>Hand Calculation</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>-------------</td>
<td>------------------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>VB&lt;sub&gt;x&lt;/sub&gt;</td>
<td>32.4</td>
<td>32.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>VB&lt;sub&gt;y&lt;/sub&gt;</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Min base shear (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>VB&lt;sub&gt;x&lt;/sub&gt;</td>
<td>7.2</td>
<td>7.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>VB&lt;sub&gt;y&lt;/sub&gt;</td>
<td>7.2</td>
<td>7.2</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2016\IS 1893 2016 Static Seismic.STD is typically installed with the program.

**STAAD SPACE**

START JOB INFORMATION
ENGINEER DATE 26-Jun-18
END JOB INFORMATION
INPUT WIDTH 79
*SET STAR 0
UNIT METER KN

JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 4 3 0; 4 4 0 0; 5 0 0 4; 6 0 3 4; 7 4 3 4; 8 4 0 4; 9 8 3 0; 10 8 0 0; 11 8 3 4; 12 8 0 4; 13 12 3 0; 14 12 0 0; 15 12 3 4; 16 12 0 4; 17 16 3 0; 18 16 0 0; 19 16 3 4; 20 16 0 4; 21 0 0 8; 22 0 3 8; 23 4 3 8; 24 4 0 8; 25 8 3 8; 26 8 0 8; 27 12 3 8; 28 12 0 8; 29 16 3 8; 30 16 0 8; 31 0 0 12; 32 0 3 12; 33 4 3 12; 34 4 0 12; 35 8 3 12; 36 8 0 12; 37 12 3 12; 38 12 0 12; 39 16 3 12; 40 16 0 12; 41 0 6 0; 42 4 6 0; 43 0 6 4; 44 4 6 4; 45 8 6 0; 46 8 6 4; 47 12 6 0; 48 12 6 4; 49 16 6 0; 50 16 6 4; 51 0 6 8; 52 4 6 8; 53 8 6 8; 54 12 6 8; 55 16 6 8; 56 0 6 12; 57 4 6 12; 58 8 6 12; 59 12 6 12; 60 16 6 12; 61 0 9 0; 62 4 9 0; 63 0 9 4; 64 4 9 4; 65 8 9 0; 66 8 9 4; 67 12 9 0; 68 12 9 4; 69 16 9 0; 70 16 9 4; 71 0 9 8; 72 4 9 8; 73 8 9 8; 74 12 9 8; 75 16 9 8; 76 0 9 12; 77 4 9 12; 78 8 9 12; 79 12 9 12; 80 16 9 12;

MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 3 9; 10 7 11; 11 9 10; 12 9 11; 13 11 12; 14 9 13; 15 11 15; 16 13 14; 17 13 15; 18 15 16; 19 13 17; 20 15 19; 21 17 18; 22 17 19; 23 19 28; 24 6 22; 25 7 23; 26 11 25; 27 15 27; 28 19 29; 29 21 22; 30 22 23; 31 23 24; 32 23 25; 33 25 26; 34 25 27; 35 27 28; 36 27 29; 37 29 30; 38 22 32; 39 23 33; 40 25 35; 41 27 37; 42 29 39; 43 31 32; 44 32 33; 45 33 34; 46 33 35; 47 35 36; 48 35 37; 49 37 38; 50 37 39; 51 39 40; 52 2 41; 53 3 42; 54 6 43; 55 7 44; 56 9 45; 57 11 46; 58 13 47; 59 15 48; 60 17 49; 61 19 50; 62 22 51; 63 23 52; 64 25 53; 65 27 54; 66 29 55; 67 32 56; 68 33 57; 69 35 58; 70 37 59; 71 39 60; 72 41 42; 73 41 43; 74 42 44; 75 43 44; 76 42 45; 77 44 46; 78 45 46; 79 45 47; 80 46 48; 81 47 48; 82 47 49; 83 48 50; 84 49 50; 85 43 51; 86 44 52; 87 46 53; 88 48 54; 89 50 55; 90 51 52; 91 52 53; 92 53 54; 93 54 55; 94 51 56; 95 52 57; 96 53 58; 97 54 59; 98 55
DEFINE MATERIAL ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL

MEMBER PROPERTY AMERICAN
1 TO 153 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 10 12 14 16 18 20 21 24 26 28 30 31 34 36 38 40 FIXED
*DEFINE 1893 LOAD
DEFINE IS1893 2016 LOAD
*ZONE 0.36 RF 5 I 1.2 SS 1 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1.2 SS 1 ST 1 DM 0.05
ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05
ZONE 0.36 RF 5 I 1.2 SS 3 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1.2 SS 3 ST 1 DM 0.05
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic TITLE SS_(+Z)
1893 LOAD Z 1
LOAD 3 LOADTYPE Seismic TITLE SS_(+Y)
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH

STAAD Output
BASE SHEAR AND TIME PERIOD IN X
*********************************************************
* UNITS - KN    METE                                    *
* TIME PERIOD FOR X 1893 LOADING = 0.38971 SEC        *
* SA/G PER 1893= 2.500, LOAD FACTOR= 1.000         *
* VB PER 1893= 0.1080 X 300.00= 32.40 KN        *
* VB Act Based on Clause 7.2.1 = 32.40 KN         *
* VB Min based on Clause 7.2.2 = 7.20 KN        *
* *********************************************************
**NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6 base shear and time period in Z

****************************
* UNITS - KN METE *
* TIME PERIOD FOR Z 1893 LOADING = 0.38971 SEC *
* SA/G PER 1893= 2.500, LOAD FACTOR= 1.000 *
* VB PER 1893= 0.1000 X 300.00= 32.40 KN *
* VB Act Based on Clause 7.2.1 = 32.40 KN *
* VB Min based on Clause 7.2.2 = 7.20 KN *
* 
****************************

**NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6 base shear and time period in Y

****************************
* UNITS - KN METE *
* Calculation of SA/G for Y based on Clause 6.4.6 *
* SA/G PER 1893= 2.500, LOAD FACTOR= 1.000 *
* VB PER 1893= 0.0720 X 300.00= 21.60 KN *
* 
****************************

STAAD SPACE -- PAGE NO.

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN )</th>
<th>TORSIONAL LOAD - 1</th>
<th>LOAD - 1</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>LOAD (KN )</td>
<td>MOMENT (KN -METE) FACTOR - 1.000</td>
</tr>
<tr>
<td>-------</td>
<td>---------------------</td>
<td>--------------------</td>
<td>----------</td>
</tr>
<tr>
<td>39</td>
<td>FX</td>
<td>2.314 MY</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 2.314</td>
<td>0.000 AT LEVEL 3.000 METE</td>
</tr>
<tr>
<td></td>
<td></td>
<td>VB PER 1893 = 32.400 KN</td>
<td></td>
</tr>
<tr>
<td>60</td>
<td>FX</td>
<td>9.257 MY</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 9.257</td>
<td>0.000 AT LEVEL 6.000 METE</td>
</tr>
<tr>
<td></td>
<td></td>
<td>VB PER 1893 = 32.400 KN</td>
<td></td>
</tr>
<tr>
<td>80</td>
<td>FX</td>
<td>20.829 MY</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 20.829</td>
<td>0.000 AT LEVEL 9.000 METE</td>
</tr>
<tr>
<td></td>
<td></td>
<td>VB PER 1893 = 32.400 KN</td>
<td></td>
</tr>
</tbody>
</table>

STAAD SPACE -- PAGE NO.

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN )</th>
<th>TORSIONAL LOAD - 2</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>LOAD (KN )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>MOMENT (KN -METE) FACTOR - 1.000</td>
</tr>
<tr>
<td>-------</td>
<td>---------------------</td>
<td>--------------------</td>
</tr>
<tr>
<td>39</td>
<td>FZ</td>
<td>2.314 MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 2.314</td>
</tr>
<tr>
<td></td>
<td></td>
<td>VB PER 1893 = 32.400 KN</td>
</tr>
<tr>
<td>60</td>
<td>FZ</td>
<td>9.257 MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 9.257</td>
</tr>
<tr>
<td></td>
<td></td>
<td>VB PER 1893 = 32.400 KN</td>
</tr>
<tr>
<td>80</td>
<td>FZ</td>
<td>20.829 MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TOTAL = 20.829</td>
</tr>
<tr>
<td></td>
<td></td>
<td>VB PER 1893 = 32.400 KN</td>
</tr>
</tbody>
</table>

Verification Examples
V.06 Loading
## Verification Examples

V.06 Loading

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN)</th>
<th>TORSIONAL MOMENT (KN-METE)</th>
<th>FACTOR</th>
<th>LOAD - 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>1.000</td>
<td>----------</td>
</tr>
<tr>
<td>39</td>
<td>FY</td>
<td>1.543</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>TOTAL</td>
<td>1.543</td>
<td>0.000</td>
<td>AT LEVEL 3.000 METE</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>21.600 KN</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>60</td>
<td>FY</td>
<td>6.171</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>TOTAL</td>
<td>6.171</td>
<td>0.000</td>
<td>AT LEVEL 6.000 METE</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>21.600 KN</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>80</td>
<td>FY</td>
<td>13.886</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>TOTAL</td>
<td>13.886</td>
<td>0.000</td>
<td>AT LEVEL 9.000 METE</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>21.600 KN</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Related Links
- **TR.31.2.10 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load** (on page 2582)

### V. IS 1893 2016 GL Calculation


### Details

The following structure is modelled.
The STAAD.Pro load definition parameters are also specified:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>STAAD.Pro Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seismic zone factor, Z</td>
<td>0.16</td>
<td>Z 0.16</td>
</tr>
<tr>
<td>Importance factor, I</td>
<td>1</td>
<td>I 1</td>
</tr>
<tr>
<td>Response reduce factor, R</td>
<td>3 (OMRF)</td>
<td>RF 3</td>
</tr>
<tr>
<td>Soil site condition</td>
<td>Hard soil</td>
<td>SS 1</td>
</tr>
</tbody>
</table>
Verification Examples
V.06 Loading

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>STAAD.Pro Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structure type</td>
<td>Steel moment-resisting frame building</td>
<td>ST 3</td>
</tr>
</tbody>
</table>

Total height of the building is 100 ft
Dimension of each floor slab is 20 ft × 10 ft
Floor to floor height for super structure is 10 ft
Floor to floor height for sub structure is 5 ft
GL is located at -2 ft (Y co-ordinate)
Depth of Foundation (DT) is (20-2) = 18 ft = 5.4864 m
All the analytical members are W shaped steel beam (W12X190)
Supports are fixed
The generated model is subjected to joint weight(100kip). The base shear and its distribution along the height is calculated in Global X, Global Z, Global Y direction.

Verification
Design horizontal acceleration coefficient value calculation for above GL and below GL for Global X(Ah), Global Z(Ah), Global Y(Av) direction.

Table 453: Calculated values for hard soil

<table>
<thead>
<tr>
<th>Value</th>
<th>X</th>
<th>Z</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time Period (s)</td>
<td>1.103</td>
<td>1.103</td>
<td></td>
</tr>
<tr>
<td>Sa/g</td>
<td>0.907</td>
<td>0.907</td>
<td>2.5</td>
</tr>
<tr>
<td>(1/T, 0.4s &lt; t &lt; 4s)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>A_h or A_v</td>
<td>0.02418</td>
<td>0.02418</td>
<td>0.04445</td>
</tr>
<tr>
<td>A_h or A_v 30 m</td>
<td>0.01209</td>
<td>0.01209</td>
<td>0.0222</td>
</tr>
<tr>
<td>Reduced A_h or A_v</td>
<td>0.02197</td>
<td>0.02197</td>
<td>0.04038</td>
</tr>
</tbody>
</table>

Base shear calculation and its distribution along the height for above GL and below GL for Global X, Global Z, Global Y direction and V_{bmin} calculation

Table 454: Vertical distribution of forces above GL

<table>
<thead>
<tr>
<th>Story Level in ft from base.</th>
<th>Wi (kip)</th>
<th>hi (m from GL)</th>
<th>Wi x hi^2</th>
<th>(Wi x hi^2)/Σ (Wi x hi^2)</th>
<th>Qx (kip)</th>
<th>Qz (kip)</th>
<th>Qy (kip)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>400</td>
<td>24.9936</td>
<td>2,689,600</td>
<td>0.3074</td>
<td>26.761</td>
<td>26.761</td>
<td>49.179</td>
</tr>
</tbody>
</table>
### Table 455: Vertical distribution of forces below GL

<table>
<thead>
<tr>
<th>Story Level in ft from base.</th>
<th>Wi (kip)</th>
<th>hi (m from GL)</th>
<th>Wi× hi²</th>
<th>(Wi × hi²)/Σ (Wi × hi²)</th>
<th>Qx (kip)</th>
<th>Qz (kip)</th>
<th>Qy (kip)</th>
</tr>
</thead>
<tbody>
<tr>
<td>90</td>
<td>400</td>
<td>21.9456</td>
<td>2,073,600</td>
<td>0.2370</td>
<td>20.632</td>
<td>20.632</td>
<td>37.916</td>
</tr>
<tr>
<td>80</td>
<td>400</td>
<td>18.8976</td>
<td>1,537,600</td>
<td>0.1757</td>
<td>15.299</td>
<td>15.299</td>
<td>28.115</td>
</tr>
<tr>
<td>70</td>
<td>400</td>
<td>15.8496</td>
<td>1,081,600</td>
<td>0.1236</td>
<td>10.762</td>
<td>10.762</td>
<td>19.777</td>
</tr>
<tr>
<td>60</td>
<td>400</td>
<td>12.8016</td>
<td>705,600</td>
<td>0.0806</td>
<td>7.021</td>
<td>7.021</td>
<td>12.902</td>
</tr>
<tr>
<td>50</td>
<td>400</td>
<td>9.7536</td>
<td>409,600</td>
<td>0.0468</td>
<td>4.075</td>
<td>4.075</td>
<td>7.489</td>
</tr>
<tr>
<td>40</td>
<td>400</td>
<td>6.7056</td>
<td>193,600</td>
<td>0.0221</td>
<td>1.926</td>
<td>1.926</td>
<td>3.540</td>
</tr>
<tr>
<td>30</td>
<td>400</td>
<td>3.6576</td>
<td>57,600</td>
<td>0.0066</td>
<td>0.573</td>
<td>0.573</td>
<td>1.053</td>
</tr>
<tr>
<td>20</td>
<td>400</td>
<td>0.6096</td>
<td>1,600</td>
<td>0.0002</td>
<td>0.016</td>
<td>0.016</td>
<td>0.029</td>
</tr>
<tr>
<td>Summation</td>
<td>3,600</td>
<td>8,750,400</td>
<td>1</td>
<td></td>
<td>87.064</td>
<td>87.064</td>
<td>160</td>
</tr>
<tr>
<td>Vbh</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>87.064</td>
<td>87.064</td>
<td>160</td>
</tr>
<tr>
<td>Vbmin</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>39.6</td>
<td>39.6</td>
<td></td>
</tr>
</tbody>
</table>

Verification Examples

V.06 Loading

STAAD.Pro 3500 User Manual
## Results

### Table 456: Comparison of results

<table>
<thead>
<tr>
<th>Location</th>
<th>Shear values</th>
<th>Hand Calculations</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Qi(X)</td>
<td>Qi(Z)</td>
<td>Qi(Y)</td>
<td>Qi(X)</td>
</tr>
<tr>
<td>Above GL</td>
<td>Story shear (kip)</td>
<td>26.761</td>
<td>26.761</td>
<td>49.179</td>
<td>26.761</td>
</tr>
<tr>
<td></td>
<td></td>
<td>20.632</td>
<td>20.632</td>
<td>37.916</td>
<td>20.632</td>
</tr>
<tr>
<td></td>
<td></td>
<td>15.299</td>
<td>15.299</td>
<td>28.115</td>
<td>15.299</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7.021</td>
<td>7.021</td>
<td>12.902</td>
<td>7.021</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4.075</td>
<td>4.075</td>
<td>7.489</td>
<td>4.075</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.926</td>
<td>1.926</td>
<td>3.540</td>
<td>1.962</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.573</td>
<td>0.573</td>
<td>1.053</td>
<td>0.573</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.016</td>
<td>0.016</td>
<td>0.029</td>
<td>0.016</td>
</tr>
<tr>
<td>Base Shear (kip)</td>
<td></td>
<td>87.064</td>
<td>87.064</td>
<td>160</td>
<td>87.064</td>
</tr>
<tr>
<td></td>
<td>Base Shear (kip)</td>
<td>17.579</td>
<td>17.579</td>
<td>32.304</td>
<td>17.579</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro Input File**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2016\IS 1893 2016 GL Calculation.STD` is typically installed with the program.

```plaintext
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Aug-18
END JOB INFORMATION
UNIT FEET KIP
```
JOINT COORDINATES
1 0 -20 0; 2 0 -10 0; 3 20 -20 0; 4 20 -10 0; 5 0 0 0; 6 20 0 0;
7 0 -10 10; 8 20 -10 10; 9 0 0 10; 10 20 0 10; 11 0 -20 10;
12 20 -20 10; 13 0 10 0; 14 20 10 0; 15 0 0 10; 16 20 10 0; 17 0 20 0;
18 20 20 0; 19 0 20 10; 20 20 20 10; 21 0 30 0; 22 20 30 0; 23 0 30 10;
24 20 30 10; 25 0 40 0; 26 20 40 0; 27 0 50 0; 28 20 50 0; 29 0 50 10;
30 20 50 10; 31 0 60 0; 32 20 60 0; 33 0 60 10; 34 20 60 10; 35 0 60 10;
36 20 60 10; 37 0 70 0; 38 20 70 0; 39 0 70 10; 40 20 70 10; 41 0 80 0;
42 20 80 0; 43 0 80 10; 44 20 80 10; 45 0 -15 10; 46 0 -15 0;
47 20 -15 10; 48 20 -15 0;
MEMBER INCIDENCES
1 1 46; 2 3 48; 3 2 4; 4 4 6; 5 2 4; 6 5 6; 7 2 7; 8 4 8; 9 5 9;
10 6 10; 11 11 45; 12 12 47; 13 7 9; 14 8 10; 15 7 8; 16 9 10; 17 5 13;
18 6 14; 19 13 14; 20 13 15; 21 14 16; 22 9 15; 23 10 16; 24 15 16;
25 13 17; 26 14 18; 27 17 18; 28 17 19; 29 18 20; 30 15 19; 31 16 20;
32 19 20; 33 17 21; 34 18 22; 35 21 22; 36 21 23; 37 22 24; 38 19 23;
39 20 24; 40 23 24; 41 21 25; 42 22 26; 43 25 26; 44 25 27; 45 26 28;
46 23 27; 47 24 28; 48 27 28; 49 25 29; 50 26 30; 51 29 30; 52 29 31;
53 30 31; 54 27 31; 55 28 32; 56 31 32; 57 29 33; 58 30 34; 59 33 34;
60 31 35; 61 32 36; 62 35 36; 63 33 37; 64 34 38; 65 37 38; 66 35 39;
67 36 40; 68 39 40; 69 33 35; 70 34 36; 71 37 39; 72 38 40; 73 37 41;
74 38 42; 75 41 42; 76 39 43; 77 40 44; 78 43 44; 79 41 43; 80 42 44;
81 45 7; 82 46 2; 83 47 8; 84 48 4; 85 45 47; 86 47 48; 87 46 46;
DEFINITE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 88 TABLE ST W12X190
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 3 11 12 FIXED
DEFINE IS1893 2016 LOAD
ZONE 0.16 RF 3 I 1 SS 1 ST 3 GL -2
JOINT WEIGHT
2 4 TO 10 13 TO 48 WEIGHT 100
LOAD 1 LOADTYPE Seismic TITLE SS_ (+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic TITLE SS_ (+Z)
1893 LOAD Z 1
LOAD 3 LOADTYPE Seismic TITLE SS_ (+Y)
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
FINISH

**NOTE: Equivalent Static Analysis should preferably be performed for regular**
structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6

BASE SHEAR AND TIME PERIOD IN X

--ABOVE GROUND LEVEL

*********************************************************
* UNITS - KIP   FEET                                    *
* TIME PERIOD FOR X 1893 LOADING = 1.10263 SEC           *
* SA/G PER 1893= 0.907, LOAD FACTOR= 1.000              *
* VB PER 1893= 0.0242 X 3600.00= 87.06 KIP              *
* VB Act Based on Clause 7.2.1 = 87.06 KIP              *
* VB Min based on Clause 7.2.2 = 39.60 KIP              *
*********************************************************

--BELOW GROUND LEVEL

*********************************************************
* UNITS - KIP   FEET                                    *
* TIME PERIOD FOR X 1893 LOADING = 1.10263 SEC           *
* SA/G PER 1893= 0.907, LOAD FACTOR= 1.000              *
* FACTOR V PER 1893 AT GL= 0.0242 X 800.00               *
* FACTOR V PER 1893 AT 30 M= 0.0121 X 800.00             *
* FACTOR V PER 1893= 0.0220 X 800.00                    *
* VB Min based on Clause 7.2.2 = 8.80 KIP               *
*********************************************************

--MASS AND BASE SHEAR SUMMERY

Units:KIP FEET

<table>
<thead>
<tr>
<th>Category</th>
<th>Mass</th>
<th>Ah</th>
<th>VB Calculated</th>
<th>VB Minimum</th>
<th>VB Final</th>
</tr>
</thead>
<tbody>
<tr>
<td>Above GL</td>
<td>3600.00000</td>
<td>0.0241846</td>
<td>87.06450</td>
<td>39.60000</td>
<td>87.06450</td>
</tr>
<tr>
<td>Below GL</td>
<td>800.00000</td>
<td>0.0219731</td>
<td>17.57852</td>
<td>8.80000</td>
<td>17.57852</td>
</tr>
</tbody>
</table>

***NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6

BASE SHEAR AND TIME PERIOD IN Z

--ABOVE GROUND LEVEL

*********************************************************
* UNITS - KIP   FEET                                    *
* TIME PERIOD FOR Z 1893 LOADING = 1.10263 SEC           *
* SA/G PER 1893= 0.907, LOAD FACTOR= 1.000              *
* VB PER 1893= 0.0242 X 3600.00= 87.06 KIP              *
* VB Act Based on Clause 7.2.1 = 87.06 KIP              *
* VB Min based on Clause 7.2.2 = 39.60 KIP              *
*********************************************************

--BELOW GROUND LEVEL

*********************************************************
**Verification Examples**

**V.06 Loading**

```
* TIME PERIOD FOR Z 1893 LOADING = 1.10263 SEC *
* SA/G PER 1893= 0.907, LOAD FACTOR= 1.000 *
* FACTOR V PER 1893 AT GL= 0.0242 X 800.00 *
* FACTOR V PER 1893 AT 30 M= 0.0121 X 800.00 *
* FACTOR V PER 1893= 0.0220 X 800.00 *
* VB Min based on Clause 7.2.2 = 8.80 KIP *

*********************************************************
--MASS AND BASE SHEAR SUMMERY
Units: KIP FEET
<table>
<thead>
<tr>
<th>Category</th>
<th>Mass</th>
<th>Ah</th>
<th>VB Calculated</th>
<th>VB Minimum</th>
<th>VB Final</th>
</tr>
</thead>
<tbody>
<tr>
<td>Above GL</td>
<td>3600.00000</td>
<td>0.0241846</td>
<td>87.06450</td>
<td>39.60000</td>
<td></td>
</tr>
<tr>
<td>Below GL</td>
<td>800.00000</td>
<td>0.0219731</td>
<td>17.57852</td>
<td>8.80000</td>
<td></td>
</tr>
</tbody>
</table>

***NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6 BASE SHEAR AND TIME PERIOD IN Y
--ABOVE GROUND LEVEL
*********************************************************
--BELOW GROUND LEVEL
*********************************************************
```

---

**STAAD.Pro 3504 User Manual**
Below GL  |   800.0000  |  0.0403805|  32.30436 |   0.00000|  
| 32.30436|  

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT</td>
<td>LATERAL</td>
<td>TORSIONAL</td>
<td>LOAD -</td>
</tr>
<tr>
<td>-------</td>
<td>------------</td>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>LOAD (KIP )</td>
<td>MOMENT (KIP -FEET)</td>
<td>FACTOR - 1.000</td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>------------</td>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>45</td>
<td>FX</td>
<td>0.879</td>
<td>MY</td>
</tr>
<tr>
<td>46</td>
<td>FX</td>
<td>0.879</td>
<td>MY</td>
</tr>
<tr>
<td>47</td>
<td>FX</td>
<td>0.879</td>
<td>MY</td>
</tr>
<tr>
<td>48</td>
<td>FX</td>
<td>0.879</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL</td>
<td>3.516</td>
<td>0.000</td>
<td>AT LEVEL</td>
</tr>
<tr>
<td>VB PER 1893 = 17.579 KIP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>FX</td>
<td>3.516</td>
<td>MY</td>
</tr>
<tr>
<td>4</td>
<td>FX</td>
<td>3.516</td>
<td>MY</td>
</tr>
<tr>
<td>7</td>
<td>FX</td>
<td>3.516</td>
<td>MY</td>
</tr>
<tr>
<td>8</td>
<td>FX</td>
<td>3.516</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL</td>
<td>14.063</td>
<td>0.000</td>
<td>AT LEVEL</td>
</tr>
<tr>
<td>VB PER 1893 = 17.579 KIP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>FX</td>
<td>0.004</td>
<td>MY</td>
</tr>
<tr>
<td>6</td>
<td>FX</td>
<td>0.004</td>
<td>MY</td>
</tr>
<tr>
<td>9</td>
<td>FX</td>
<td>0.004</td>
<td>MY</td>
</tr>
<tr>
<td>10</td>
<td>FX</td>
<td>0.004</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL</td>
<td>0.016</td>
<td>0.000</td>
<td>AT LEVEL</td>
</tr>
<tr>
<td>VB PER 1893 = 87.064 KIP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>FX</td>
<td>0.143</td>
<td>MY</td>
</tr>
<tr>
<td>14</td>
<td>FX</td>
<td>0.143</td>
<td>MY</td>
</tr>
<tr>
<td>15</td>
<td>FX</td>
<td>0.143</td>
<td>MY</td>
</tr>
<tr>
<td>16</td>
<td>FX</td>
<td>0.143</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL</td>
<td>0.573</td>
<td>0.000</td>
<td>AT LEVEL</td>
</tr>
<tr>
<td>VB PER 1893 = 87.064 KIP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>FX</td>
<td>0.482</td>
<td>MY</td>
</tr>
<tr>
<td>18</td>
<td>FX</td>
<td>0.482</td>
<td>MY</td>
</tr>
<tr>
<td>19</td>
<td>FX</td>
<td>0.482</td>
<td>MY</td>
</tr>
<tr>
<td>20</td>
<td>FX</td>
<td>0.482</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL</td>
<td>1.926</td>
<td>0.000</td>
<td>AT LEVEL</td>
</tr>
<tr>
<td>VB PER 1893 = 87.064 KIP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>FX</td>
<td>1.019</td>
<td>MY</td>
</tr>
<tr>
<td>22</td>
<td>FX</td>
<td>1.019</td>
<td>MY</td>
</tr>
<tr>
<td>23</td>
<td>FX</td>
<td>1.019</td>
<td>MY</td>
</tr>
<tr>
<td>24</td>
<td>FX</td>
<td>1.019</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOTAL</td>
<td>4.075</td>
<td>0.000</td>
<td>AT LEVEL</td>
</tr>
<tr>
<td>VB PER 1893 = 87.064 KIP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>FX</td>
<td>1.755</td>
<td>MY</td>
</tr>
<tr>
<td>26</td>
<td>FX</td>
<td>1.755</td>
<td>MY</td>
</tr>
<tr>
<td>27</td>
<td>FX</td>
<td>1.755</td>
<td>MY</td>
</tr>
<tr>
<td>28</td>
<td>FX</td>
<td>1.755</td>
<td>MY</td>
</tr>
<tr>
<td>STAAD SPACE</td>
<td>-- PAGE NO.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

STAAD.Pro 3505 User Manual
### Verification Examples

#### V.06 Loading

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>FZ</td>
<td>3.516</td>
<td>0.879</td>
</tr>
<tr>
<td>4</td>
<td>FZ</td>
<td>3.516</td>
<td>0.879</td>
</tr>
<tr>
<td>7</td>
<td>FZ</td>
<td>3.516</td>
<td>0.879</td>
</tr>
<tr>
<td>8</td>
<td>FZ</td>
<td>3.516</td>
<td>0.879</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>9</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>10</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>FZ</td>
<td>0.143</td>
<td>0.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>19</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>21</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>23</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>25</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>27</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>29</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>31</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>33</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>35</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>37</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>39</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>41</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>43</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>45</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>47</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>49</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>51</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>53</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>55</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>57</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>59</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>61</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>63</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>65</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>67</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>69</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>71</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>73</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>75</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>77</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>79</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>81</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>83</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>85</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>87</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>89</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>91</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>93</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral</th>
<th>Torsional Load</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>95</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>97</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>99</td>
<td>FZ</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>STAAD SPACE</td>
<td>-- PAGE NO.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------------</td>
<td>-------------</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Verification Examples**

**V.06 Loading**

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 10,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 20,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 30,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 40,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 50,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 60,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 70,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

### Loading at Level 80,000 Feet

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

---

**STAAD Pro 3507 User Manual**
<table>
<thead>
<tr>
<th>LOAD (KIP )</th>
<th>MOMENT (KIP -FEET)</th>
<th>FACTOR - 1.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>45 FY</td>
<td>1.615 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>46 FY</td>
<td>1.615 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>47 FY</td>
<td>1.615 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>48 FY</td>
<td>1.615 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td>6.461</td>
<td>0.000 AT LEVEL -15.000 FEET</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>32.304 KIP</td>
<td></td>
</tr>
<tr>
<td>2 FY</td>
<td>6.461 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>4 FY</td>
<td>6.461 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>7 FY</td>
<td>6.461 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>8 FY</td>
<td>6.461 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td>25.843</td>
<td>0.000 AT LEVEL -10.000 FEET</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>32.304 KIP</td>
<td></td>
</tr>
<tr>
<td>5 FY</td>
<td>0.007 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>6 FY</td>
<td>0.007 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>9 FY</td>
<td>0.007 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>10 FY</td>
<td>0.007 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td>0.029</td>
<td>0.000 AT LEVEL 0.000 FEET</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>160.000 KIP</td>
<td></td>
</tr>
<tr>
<td>13 FY</td>
<td>0.263 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>14 FY</td>
<td>0.263 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>15 FY</td>
<td>0.263 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>16 FY</td>
<td>0.263 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td>1.053</td>
<td>0.000 AT LEVEL 10.000 FEET</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>160.000 KIP</td>
<td></td>
</tr>
<tr>
<td>17 FY</td>
<td>0.885 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>18 FY</td>
<td>0.885 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>19 FY</td>
<td>0.885 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>20 FY</td>
<td>0.885 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td>3.540</td>
<td>0.000 AT LEVEL 20.000 FEET</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>160.000 KIP</td>
<td></td>
</tr>
<tr>
<td>21 FY</td>
<td>1.872 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>22 FY</td>
<td>1.872 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>23 FY</td>
<td>1.872 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>24 FY</td>
<td>1.872 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td>7.489</td>
<td>0.000 AT LEVEL 30.000 FEET</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>160.000 KIP</td>
<td></td>
</tr>
<tr>
<td>25 FY</td>
<td>3.225 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>26 FY</td>
<td>3.225 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>27 FY</td>
<td>3.225 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>28 FY</td>
<td>3.225 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td>12.902</td>
<td>0.000 AT LEVEL 40.000 FEET</td>
</tr>
<tr>
<td>VB PER 1893</td>
<td>160.000 KIP</td>
<td></td>
</tr>
<tr>
<td>29 FY</td>
<td>4.944 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>30 FY</td>
<td>4.944 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>31 FY</td>
<td>4.944 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>32 FY</td>
<td>4.944 MY</td>
<td>0.000</td>
</tr>
<tr>
<td>TOTAL</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Story</th>
<th>FY</th>
<th>MY</th>
<th>VB</th>
</tr>
</thead>
<tbody>
<tr>
<td>33</td>
<td>7.029</td>
<td>0.000</td>
<td>19.777</td>
</tr>
<tr>
<td>34</td>
<td>7.029</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>7.029</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>7.029</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td><strong>Total</strong> = 19.777</td>
</tr>
</tbody>
</table>

**VB PER 1893 = 160.000 KIP**

<table>
<thead>
<tr>
<th>Story</th>
<th>FY</th>
<th>MY</th>
<th>VB</th>
</tr>
</thead>
<tbody>
<tr>
<td>37</td>
<td>9.479</td>
<td>0.000</td>
<td>28.115</td>
</tr>
<tr>
<td>38</td>
<td>9.479</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>9.479</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>9.479</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td><strong>Total</strong> = 28.115</td>
</tr>
</tbody>
</table>

**VB PER 1893 = 160.000 KIP**

<table>
<thead>
<tr>
<th>Story</th>
<th>FY</th>
<th>MY</th>
<th>VB</th>
</tr>
</thead>
<tbody>
<tr>
<td>41</td>
<td>12.295</td>
<td>0.000</td>
<td>37.916</td>
</tr>
<tr>
<td>42</td>
<td>12.295</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>43</td>
<td>12.295</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>44</td>
<td>12.295</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td><strong>Total</strong> = 37.916</td>
</tr>
</tbody>
</table>

**STAAD SPACE**

<table>
<thead>
<tr>
<th>Story</th>
<th>FY</th>
<th>MY</th>
<th>VB</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td></td>
<td></td>
<td>49.179</td>
</tr>
</tbody>
</table>

**VB PER 1893 = 160.000 KIP**

**Related Links**

- [TR.28.2.2 Check Irregularities](on page 2531)

**V. IS 1893 2016 Irregular Modes of Oscillation**

Verify irregular modes of oscillation check in accordance to IS 1893 2016 Part 1 specifications.

**Details**

A three story structure is modelled with floor diaphragms defined at the story levels.
Seismic parameters:

- Seismic Zone Factor, $Z = 0.36$
- Importance Factor, $I = 1.2$
- Response Reduction Factor, $R = 5$
- Site soil conditions: hard soil (SS 1)
- Structure type: other (ST 5)

**Verification**

Since the structure is in Zone V the check has two parts. These include:

1. First three modes together contribute at least 65% mass participation factor in each principal plan direction.
2. The fundamental lateral natural periods of the building in the two principal plan directions are away from each other by at least 10% of the larger value.

Looking at the mass participation factor table in output file following observations are made:
Also, it is observed from mass participation factor table the maximum mass participation for X direction occurs in mode 1 and for Z direction occurs in mode 2. Hence for the second part of the check the period of these modes needs to be checked for being away from each other by at least 10% of the larger value.

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Mass Participation X</th>
<th>Mass Participation Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>70.14</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>91.38</td>
</tr>
<tr>
<td>3</td>
<td>21.22</td>
<td>0</td>
</tr>
<tr>
<td>Summation</td>
<td>91.36</td>
<td>91.38</td>
</tr>
<tr>
<td>Status (FAIL if summation &lt; 65)</td>
<td>OK</td>
<td>OK</td>
</tr>
</tbody>
</table>

Results

Table 457: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Participation Summation X (%)</td>
<td>91.36</td>
<td>91.359</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Mass Participation Summation Z(%)</td>
<td>91.38</td>
<td>91.382</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Check status</td>
<td>OK</td>
<td>OK</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Percent separation</td>
<td>6.7574</td>
<td>6.7574</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Check status</td>
<td>Failed</td>
<td>Failed</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2016\IS 1893 2016 Irregular Modes of Oscillation.STD is typically installed with the program.
**STAAD.Pro Output**

***NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6 BASE SHEAR AND TIME PERIOD IN X***
**UNITS - KN METE**

* TIME PERIOD FOR X 1893 LOADING = 0.60374 SEC *
* SA/G PER 1893= 1.656, LOAD FACTOR= 1.000 *
* VB PER 1893= 0.0716 X 850.41= 60.85 KN *
* VB Act Based on Clause 7.2.1 = 60.85 KN *
* VB Min based on Clause 7.2.2 = 20.41 KN *

**EIGEN METHOD : SUBSPACE**

---

NUMBER OF MODES REQUESTED = 6
NUMBER OF EXISTING MASSES IN THE MODEL = 21
NUMBER OF MODES THAT WILL BE USED = 6

*** EIGENSOLUTION : ADVANCED METHOD ***

**1893 RESPONSE SPECTRUM LOAD**

**RESPONSE LOAD CASE 2**

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.086</td>
<td>11.66562</td>
</tr>
<tr>
<td>2</td>
<td>0.092</td>
<td>10.87733</td>
</tr>
<tr>
<td>3</td>
<td>0.110</td>
<td>9.11988</td>
</tr>
<tr>
<td>4</td>
<td>0.240</td>
<td>4.16460</td>
</tr>
<tr>
<td>5</td>
<td>0.257</td>
<td>3.88372</td>
</tr>
<tr>
<td>6</td>
<td>0.307</td>
<td>3.25667</td>
</tr>
</tbody>
</table>

**CALCULATED FREQUENCIES FOR LOAD CASE 2**

<table>
<thead>
<tr>
<th>MODE</th>
<th>WEIGHT</th>
<th>GENERALIZED</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.947984E+02</td>
<td>4.193119E+02</td>
</tr>
<tr>
<td>2</td>
<td>5.799489E-30</td>
<td>5.193760E+02</td>
</tr>
<tr>
<td>3</td>
<td>1.799753E+02</td>
<td>2.961189E+02</td>
</tr>
<tr>
<td>4</td>
<td>4.897377E+01</td>
<td>4.206115E+02</td>
</tr>
<tr>
<td>5</td>
<td>2.240817E-32</td>
<td>5.211878E+02</td>
</tr>
<tr>
<td>6</td>
<td>1.491518E+01</td>
<td>2.973559E+02</td>
</tr>
</tbody>
</table>

**CQC**

MODAL COMBINATION METHOD USED.

**DYNAMIC WEIGHT X Y Z** 8.480507E+02 8.480507E+02 8.480507E+02 KN

**MISSING WEIGHT X Y Z** -9.353708E+00 -8.480504E+02 -9.353708E+00 KN

**MODAL WEIGHT X Y Z** 8.386626E+02 3.396127E-04 8.386626E+02 KN

**RESPONSE LOAD CASE 2**

<table>
<thead>
<tr>
<th>MODE</th>
<th>SPECTRAL ACCELERATION</th>
<th>DESIGN SEISMIC COEFFICIENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>1</td>
<td>0.25000</td>
<td>0.0108</td>
</tr>
<tr>
<td>2</td>
<td>0.25000</td>
<td>0.0108</td>
</tr>
<tr>
<td>3</td>
<td>0.25000</td>
<td>0.0108</td>
</tr>
<tr>
<td>4</td>
<td>0.25000</td>
<td>0.0108</td>
</tr>
<tr>
<td>5</td>
<td>0.25749</td>
<td>0.0111</td>
</tr>
<tr>
<td>6</td>
<td>0.30706</td>
<td>0.0133</td>
</tr>
</tbody>
</table>

**PEAK STORY SHEAR**

<table>
<thead>
<tr>
<th>STORY</th>
<th>LEVEL IN METE</th>
<th>PEAK STORY SHEAR IN KN</th>
</tr>
</thead>
</table>

---

**STAAD Space**

**PEAK STORY SHEAR**

<table>
<thead>
<tr>
<th>STORY</th>
<th>LEVEL IN METE</th>
<th>PEAK STORY SHEAR IN KN</th>
</tr>
</thead>
</table>

---

**VERIFICATION EXAMPLES**

V.06 Loading

---

**STAAD.Pro 3513 User Manual**
### Modal Base Actions

**FORCES IN KN**  
**LENGTH IN METE**

<table>
<thead>
<tr>
<th>MODE</th>
<th>PERIOD</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
<th>MY</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11.66</td>
<td>6.42</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>10.87</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>9.12</td>
<td>1.94</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>4.165</td>
<td>0.53</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>3.29</td>
<td>0.20</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>3.037</td>
<td>0.07</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**STAAD SPACE**  
-- PAGE NO.

### Participation Factors

**MASS**  
**PARTICIPATION FACTORS IN PERCENT**  
**BASE SHEAR IN KN**

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>70.14</td>
<td>0.00</td>
<td>0.00</td>
<td>70.137</td>
<td>0.00</td>
<td>0.00</td>
<td>6.42</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>91.38</td>
<td>70.137</td>
<td>0.00</td>
<td>91.382</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>21.22</td>
<td>0.00</td>
<td>0.00</td>
<td>91.359</td>
<td>0.00</td>
<td>91.382</td>
<td>1.94</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>5.77</td>
<td>0.00</td>
<td>0.00</td>
<td>97.134</td>
<td>0.00</td>
<td>91.382</td>
<td>0.53</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>7.52</td>
<td>97.134</td>
<td>0.00</td>
<td>98.897</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1.76</td>
<td>0.00</td>
<td>0.00</td>
<td>98.893</td>
<td>0.00</td>
<td>98.897</td>
<td>0.20</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**TOTAL SRSS SHEAR**  
6.74  
**TOTAL 10PCT SHEAR**  
6.74
<table>
<thead>
<tr>
<th>Node</th>
<th>Disp. (mm)</th>
<th>Disp. (mm)</th>
<th>Disp. (mm)</th>
<th>Disp. (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>79.02533</td>
<td>104.16057</td>
<td>91.54622</td>
<td>91.54622</td>
</tr>
<tr>
<td>2</td>
<td>158.54202</td>
<td>208.71021</td>
<td>183.34507</td>
<td>183.34507</td>
</tr>
<tr>
<td>3</td>
<td>238.33576</td>
<td>314.07252</td>
<td>275.55352</td>
<td>275.55352</td>
</tr>
</tbody>
</table>

---

**Diaphragm D X-max/min D Z-max/min Status**

<table>
<thead>
<tr>
<th>Dia.</th>
<th>D</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.3181</td>
<td>1.0000</td>
</tr>
<tr>
<td>2</td>
<td>1.3164</td>
<td>1.0000</td>
</tr>
<tr>
<td>3</td>
<td>1.3178</td>
<td>1.0000</td>
</tr>
</tbody>
</table>

---

**Mass Irregularity Check**

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Level</th>
<th>Mass (kN)</th>
<th>Above (kN)</th>
<th>Below (kN)</th>
<th>Ratio</th>
<th>Ratio</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.000</td>
<td>283.469</td>
<td>283.469</td>
<td>Base</td>
<td>1.000</td>
<td>N/A</td>
<td>OK</td>
</tr>
<tr>
<td>2</td>
<td>10.000</td>
<td>283.469</td>
<td>281.113</td>
<td>283.469</td>
<td>1.008</td>
<td>1.000</td>
<td>OK</td>
</tr>
<tr>
<td>3</td>
<td>15.000</td>
<td>281.113</td>
<td>283.469</td>
<td>N/A</td>
<td>0.992</td>
<td>0.992</td>
<td>OK</td>
</tr>
</tbody>
</table>
### Mass Participation

<table>
<thead>
<tr>
<th>Mode</th>
<th>Mass Part X (%)</th>
<th>Mass Part Z (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>70.137</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>91.382</td>
</tr>
<tr>
<td>3</td>
<td>21.222</td>
<td>0.000</td>
</tr>
</tbody>
</table>

**SUM**

|    | 91.359          | 91.382          |

### Fundamental Period

<table>
<thead>
<tr>
<th>X Direction</th>
<th>Z Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode</td>
<td>Period (s)</td>
</tr>
<tr>
<td>------</td>
<td>------------</td>
</tr>
<tr>
<td>1</td>
<td>11.6656</td>
</tr>
</tbody>
</table>

55. PRINT ANALYSIS RESULTS

ANALYSIS RESULTS

STAAD SPACE

---

### Joint Displacement (CM RADIANS)

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROtan</th>
<th>Y-ROtan</th>
<th>Z-ROtan</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>142.7736</td>
<td>0.2189</td>
<td>-17.2600</td>
<td>-0.0001</td>
<td>-0.0690</td>
<td>-0.0012</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>169.1551</td>
<td>0.1710</td>
<td>58.2987</td>
<td>0.0001</td>
<td>0.2332</td>
<td>0.0010</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>142.7736</td>
<td>-0.2189</td>
<td>17.2600</td>
<td>-0.0001</td>
<td>-0.0690</td>
<td>-0.0012</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>169.1551</td>
<td>0.1710</td>
<td>58.2987</td>
<td>0.0001</td>
<td>0.2332</td>
<td>0.0010</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>108.2535</td>
<td>0.0789</td>
<td>-17.2600</td>
<td>-0.0003</td>
<td>-0.0690</td>
<td>-0.0015</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>98.2930</td>
<td>0.0765</td>
<td>58.2987</td>
<td>0.0003</td>
<td>0.2332</td>
<td>0.0013</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>108.2535</td>
<td>-0.0789</td>
<td>17.2600</td>
<td>-0.0003</td>
<td>-0.0690</td>
<td>-0.0015</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>98.2930</td>
<td>0.0765</td>
<td>58.2987</td>
<td>0.0003</td>
<td>0.2332</td>
<td>0.0013</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>275.6922</td>
<td>0.3116</td>
<td>-33.2390</td>
<td>-0.0004</td>
<td>-0.1330</td>
<td>-0.0015</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>304.0858</td>
<td>0.2442</td>
<td>104.6761</td>
<td>0.0001</td>
<td>0.4187</td>
<td>0.0012</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>275.6922</td>
<td>-0.3116</td>
<td>33.2390</td>
<td>-0.0004</td>
<td>-0.1330</td>
<td>-0.0015</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>304.0858</td>
<td>0.2442</td>
<td>104.6761</td>
<td>0.0001</td>
<td>0.4187</td>
<td>0.0012</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>209.2142</td>
<td>0.1330</td>
<td>-33.2390</td>
<td>-0.0004</td>
<td>-0.1330</td>
<td>-0.0018</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>176.6821</td>
<td>0.1223</td>
<td>104.6761</td>
<td>0.0003</td>
<td>0.4187</td>
<td>0.0014</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>209.2142</td>
<td>-0.1330</td>
<td>33.2390</td>
<td>-0.0004</td>
<td>-0.1330</td>
<td>-0.0018</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>176.6821</td>
<td>0.1223</td>
<td>104.6761</td>
<td>0.0003</td>
<td>0.4187</td>
<td>0.0014</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>367.7505</td>
<td>0.3317</td>
<td>-44.2785</td>
<td>-0.0001</td>
<td>-0.1771</td>
<td>-0.0014</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>379.3563</td>
<td>0.2566</td>
<td>130.4985</td>
<td>0.0001</td>
<td>0.5220</td>
<td>0.0011</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>367.7505</td>
<td>-0.3317</td>
<td>44.2785</td>
<td>-0.0001</td>
<td>-0.1771</td>
<td>-0.0014</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>379.3563</td>
<td>0.2566</td>
<td>130.4985</td>
<td>0.0001</td>
<td>0.5220</td>
<td>0.0011</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>279.1936</td>
<td>0.1578</td>
<td>-44.2785</td>
<td>-0.0004</td>
<td>-0.1771</td>
<td>-0.0016</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>220.5188</td>
<td>0.1407</td>
<td>130.4985</td>
<td>0.0003</td>
<td>0.5220</td>
<td>0.0012</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>279.1936</td>
<td>-0.1578</td>
<td>44.2785</td>
<td>-0.0004</td>
<td>-0.1771</td>
<td>-0.0016</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>220.5188</td>
<td>0.1407</td>
<td>130.4985</td>
<td>0.0003</td>
<td>0.5220</td>
<td>0.0012</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>131.0756</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0690</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
### Support Reactions - Unit KN METE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-17.32</td>
<td>-91.62</td>
<td>2.10</td>
<td>5.25</td>
<td>1.80</td>
<td>43.32</td>
</tr>
<tr>
<td>2</td>
<td>20.53</td>
<td>74.27</td>
<td>7.08</td>
<td>17.71</td>
<td>6.10</td>
<td>51.35</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-17.32</td>
<td>91.62</td>
<td>-2.10</td>
<td>-5.25</td>
<td>1.80</td>
<td>43.32</td>
</tr>
<tr>
<td>2</td>
<td>20.53</td>
<td>74.27</td>
<td>7.08</td>
<td>17.71</td>
<td>6.10</td>
<td>51.35</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-13.11</td>
<td>-34.26</td>
<td>2.11</td>
<td>5.26</td>
<td>1.80</td>
<td>32.81</td>
</tr>
<tr>
<td>2</td>
<td>11.91</td>
<td>33.22</td>
<td>7.08</td>
<td>17.71</td>
<td>6.10</td>
<td>29.80</td>
<td></td>
</tr>
</tbody>
</table>

### Member End Forces - Structure Type = SPACE

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-91.62</td>
<td>17.32</td>
<td>2.10</td>
<td>1.80</td>
<td>-5.25</td>
<td>43.32</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>1</td>
<td>74.27</td>
<td>20.53</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>51.35</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>-91.62</td>
<td>17.32</td>
<td>-2.10</td>
<td>-1.80</td>
<td>-5.25</td>
<td>43.26</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>1</td>
<td>74.27</td>
<td>20.53</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>51.30</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>91.62</td>
<td>17.32</td>
<td>2.10</td>
<td>1.80</td>
<td>-5.25</td>
<td>43.26</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>3</td>
<td>-91.62</td>
<td>17.32</td>
<td>-2.10</td>
<td>-1.80</td>
<td>-5.25</td>
<td>43.32</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>1</td>
<td>74.27</td>
<td>20.53</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>51.30</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>4</td>
<td>74.27</td>
<td>20.53</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>51.35</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>4</td>
<td>0.00</td>
<td>10.72</td>
<td>0.00</td>
<td>63.07</td>
<td>-0.00</td>
<td>43.41</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0</td>
<td>0.00</td>
<td>-10.72</td>
<td>0.00</td>
<td>-63.07</td>
<td>0.00</td>
<td>10.20</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>2</td>
<td>0.00</td>
<td>13.66</td>
<td>0.00</td>
<td>53.29</td>
<td>0.00</td>
<td>41.04</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>0</td>
<td>0.00</td>
<td>13.66</td>
<td>0.00</td>
<td>53.29</td>
<td>0.00</td>
<td>31.80</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>3</td>
<td>0.00</td>
<td>-10.72</td>
<td>0.00</td>
<td>63.07</td>
<td>-0.00</td>
<td>-43.41</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>7</td>
<td>0.00</td>
<td>10.72</td>
<td>0.00</td>
<td>-63.07</td>
<td>0.00</td>
<td>-10.20</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0</td>
<td>0.00</td>
<td>13.66</td>
<td>0.00</td>
<td>53.29</td>
<td>0.00</td>
<td>41.04</td>
</tr>
<tr>
<td>7</td>
<td>3</td>
<td>7</td>
<td>0.00</td>
<td>13.66</td>
<td>0.00</td>
<td>53.29</td>
<td>0.00</td>
<td>31.80</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>5</td>
<td>-34.26</td>
<td>13.11</td>
<td>2.11</td>
<td>1.80</td>
<td>-5.26</td>
<td>32.81</td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>1</td>
<td>34.26</td>
<td>-13.11</td>
<td>-2.11</td>
<td>-1.80</td>
<td>-5.28</td>
<td>32.74</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>2</td>
<td>33.22</td>
<td>11.91</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>29.80</td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>6</td>
<td>33.22</td>
<td>11.91</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>29.74</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>6</td>
<td>0.00</td>
<td>-0.04</td>
<td>0.00</td>
<td>0.02</td>
<td>0.00</td>
<td>0.09</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>0.08</td>
</tr>
<tr>
<td>7</td>
<td>6</td>
<td>0</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>0.08</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>7</td>
<td>34.26</td>
<td>13.11</td>
<td>2.11</td>
<td>1.80</td>
<td>-5.28</td>
<td>32.74</td>
</tr>
<tr>
<td>8</td>
<td>7</td>
<td>0</td>
<td>-34.26</td>
<td>-13.11</td>
<td>-2.11</td>
<td>-1.80</td>
<td>-5.26</td>
<td>32.81</td>
</tr>
<tr>
<td>2</td>
<td>8</td>
<td>2</td>
<td>33.22</td>
<td>11.91</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>29.74</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>8</td>
<td>33.22</td>
<td>11.91</td>
<td>7.08</td>
<td>6.10</td>
<td>17.71</td>
<td>29.80</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>2</td>
<td>-43.74</td>
<td>16.07</td>
<td>1.95</td>
<td>1.67</td>
<td>-4.87</td>
<td>40.19</td>
</tr>
<tr>
<td>9</td>
<td>9</td>
<td>9</td>
<td>43.74</td>
<td>-16.07</td>
<td>-1.95</td>
<td>-1.67</td>
<td>-4.87</td>
<td>40.17</td>
</tr>
</tbody>
</table>
### Member End Forces

<table>
<thead>
<tr>
<th>Member</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL (KN)</th>
<th>SHEAR-Y (KN)</th>
<th>SHEAR-Z (KN)</th>
<th>TORSION (KN.m)</th>
<th>MOM-Y (KN.m)</th>
<th>MOM-Z (KN.m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>2</td>
<td>2</td>
<td>31.96</td>
<td>16.42</td>
<td>5.67</td>
<td>4.88</td>
<td>14.18</td>
<td>41.06</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>3</td>
<td>43.74</td>
<td>16.07</td>
<td>-1.95</td>
<td>1.67</td>
<td>4.87</td>
<td>40.19</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>3</td>
<td>31.96</td>
<td>16.42</td>
<td>5.67</td>
<td>4.88</td>
<td>14.18</td>
<td>41.06</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>1</td>
<td>31.96</td>
<td>16.42</td>
<td>5.67</td>
<td>4.88</td>
<td>14.18</td>
<td>41.06</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>6</td>
<td>-23.50</td>
<td>12.17</td>
<td>1.96</td>
<td>1.67</td>
<td>-4.91</td>
<td>30.43</td>
</tr>
<tr>
<td>11</td>
<td>3</td>
<td>9</td>
<td>23.50</td>
<td>-12.17</td>
<td>-1.96</td>
<td>-1.67</td>
<td>-4.92</td>
<td>30.41</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>20.05</td>
<td>9.52</td>
<td>5.67</td>
<td>4.88</td>
<td>14.18</td>
<td>23.79</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>6</td>
<td>20.05</td>
<td>9.52</td>
<td>5.67</td>
<td>4.88</td>
<td>14.18</td>
<td>23.79</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>7</td>
<td>20.05</td>
<td>9.52</td>
<td>5.67</td>
<td>4.88</td>
<td>14.18</td>
<td>23.79</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>7</td>
<td>20.05</td>
<td>9.52</td>
<td>5.67</td>
<td>4.88</td>
<td>14.18</td>
<td>23.79</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>9</td>
<td>0.00</td>
<td>-47.69</td>
<td>0.00</td>
<td>46.76</td>
<td>0.00</td>
<td>-119.23</td>
</tr>
<tr>
<td>10</td>
<td>9</td>
<td>0.00</td>
<td>47.69</td>
<td>0.00</td>
<td>-46.76</td>
<td>0.00</td>
<td>-119.23</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>9</td>
<td>0.00</td>
<td>12.67</td>
<td>-0.00</td>
<td>51.32</td>
<td>-0.00</td>
<td>55.01</td>
</tr>
<tr>
<td>11</td>
<td>9</td>
<td>0.00</td>
<td>12.67</td>
<td>-0.00</td>
<td>51.32</td>
<td>-0.00</td>
<td>8.36</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>10</td>
<td>0.00</td>
<td>-12.67</td>
<td>-0.00</td>
<td>51.32</td>
<td>-0.00</td>
<td>-55.01</td>
</tr>
<tr>
<td>12</td>
<td>10</td>
<td>0.00</td>
<td>12.67</td>
<td>0.00</td>
<td>-51.32</td>
<td>0.00</td>
<td>-8.36</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>0.00</td>
<td>12.02</td>
<td>0.00</td>
<td>37.10</td>
<td>0.00</td>
<td>43.88</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>10</td>
<td>0.00</td>
<td>12.02</td>
<td>0.00</td>
<td>37.10</td>
<td>0.00</td>
<td>22.17</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>11</td>
<td>0.00</td>
<td>-0.04</td>
<td>0.00</td>
<td>0.02</td>
<td>-0.00</td>
<td>-0.10</td>
</tr>
<tr>
<td>12</td>
<td>11</td>
<td>0.00</td>
<td>0.04</td>
<td>0.00</td>
<td>-0.02</td>
<td>-0.00</td>
<td>-0.07</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>11</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>0.02</td>
<td>0.00</td>
<td>0.07</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>11</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>0.02</td>
<td>0.00</td>
<td>0.07</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>9</td>
<td>-8.72</td>
<td>11.10</td>
<td>1.35</td>
<td>1.15</td>
<td>-3.37</td>
<td>27.74</td>
</tr>
<tr>
<td>13</td>
<td>9</td>
<td>-8.72</td>
<td>11.10</td>
<td>1.35</td>
<td>-1.15</td>
<td>-3.37</td>
<td>27.75</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>9</td>
<td>5.72</td>
<td>9.31</td>
<td>3.23</td>
<td>2.78</td>
<td>8.07</td>
<td>23.27</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>5.72</td>
<td>9.31</td>
<td>3.23</td>
<td>2.78</td>
<td>8.07</td>
<td>23.27</td>
<td>27.75</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>10</td>
<td>8.72</td>
<td>11.10</td>
<td>-1.35</td>
<td>1.15</td>
<td>3.37</td>
<td>27.74</td>
</tr>
<tr>
<td>14</td>
<td>10</td>
<td>8.72</td>
<td>11.10</td>
<td>-1.35</td>
<td>1.15</td>
<td>3.37</td>
<td>27.75</td>
<td></td>
</tr>
</tbody>
</table>

### Member End Forces

<table>
<thead>
<tr>
<th>Member</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL (KN)</th>
<th>SHEAR-Y (KN)</th>
<th>SHEAR-Z (KN)</th>
<th>TORSION (KN.m)</th>
<th>MOM-Y (KN.m)</th>
<th>MOM-Z (KN.m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>10</td>
<td>1</td>
<td>-10.79</td>
<td>8.40</td>
<td>1.37</td>
<td>1.15</td>
<td>-3.42</td>
<td>21.00</td>
</tr>
<tr>
<td>15</td>
<td>10.79</td>
<td>-8.40</td>
<td>-1.37</td>
<td>-1.15</td>
<td>-3.42</td>
<td>21.01</td>
<td>27.75</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>11</td>
<td>8.25</td>
<td>5.42</td>
<td>3.23</td>
<td>2.78</td>
<td>8.07</td>
<td>13.55</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>8.25</td>
<td>5.42</td>
<td>3.23</td>
<td>2.78</td>
<td>8.07</td>
<td>13.56</td>
<td>27.75</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>12</td>
<td>10.79</td>
<td>-8.40</td>
<td>1.37</td>
<td>1.15</td>
<td>3.42</td>
<td>21.00</td>
</tr>
<tr>
<td>16</td>
<td>-10.79</td>
<td>-8.40</td>
<td>1.37</td>
<td>-1.15</td>
<td>3.42</td>
<td>21.01</td>
<td>27.75</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>12</td>
<td>8.25</td>
<td>5.42</td>
<td>3.23</td>
<td>2.78</td>
<td>8.07</td>
<td>13.55</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>8.25</td>
<td>5.42</td>
<td>3.23</td>
<td>2.78</td>
<td>8.07</td>
<td>13.56</td>
<td>27.75</td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

**V.06 Loading**

| 21 | 1 | 13  | 0.00 | -19.48 | -0.00 | 46.98 | 0.00 | -48.69 |
| 14 | 0.00 | 19.48 | 0.00 | -46.98 | 0.00 | -48.69 |
| 2  | 13  | 0.00 | 13.78 | 0.00 | 34.05 | 0.00 | 34.46 |
| 14 | 0.00 | 13.78 | 0.00 | 34.05 | 0.00 | 34.46 |
| 22 | 1   | 13  | 0.00 | 10.76 | 0.00 | 20.94 | 0.00 | 50.35 |
| 15 | 0.00 | -10.76 | -0.00 | -20.94 | -0.00 | 3.44 |
| 2  | 13  | 0.00 | 8.23  | 0.00 | 13.51 | 0.00 | 36.86 |
| 15 | 0.00 | 8.23  | 0.00 | 13.51 | 0.00 | 8.08 |
| 23 | 1   | 14  | 0.00 | -10.76 | 0.00 | 20.94 | 0.00 | -50.35 |
| 16 | 0.00 | 10.76 | -0.00 | -20.94 | -0.00 | -3.44 |
| 2  | 14  | 0.00 | 8.23  | 0.00 | 13.51 | 0.00 | 36.86 |
| 16 | 0.00 | 8.23  | 0.00 | 13.51 | 0.00 | 8.08 |
| 24 | 1   | 15  | 0.00 | -0.03  | 0.00 | 0.02  | -0.00 | -0.07 |
| 16 | 0.00 | 0.03  | -0.00 | -0.02 | -0.00 | -0.07 |
| 2  | 15  | 0.00 | 0.02  | 0.00 | 0.02  | 0.00  | 0.05 |
| 16 | 0.00 | 0.02  | 0.00 | 0.02 | 0.00  | 0.05 |

************** END OF LATEST ANALYSIS RESULT ***************

**Related Links**

- **TR.28.2.2 Check Irregularities** (on page 2531)

**V. IS 1893 2016 Mass Irregularity**

Verify mass irregularity check in accordance to IS 1893 2016 Part 1 specifications.

**Details**

A three story structure is modelled with floor diaphragms defined at the story levels.
Verification Examples
V.06 Loading

Seismic parameters:

Seismic Zone Factor, Z = 0.36
Importance Factor, I = 1.2
Response Reduction Factor, R = 5
Site soil conditions: hard soil (SS 1)
Structure type: other: (ST 5)

*Verification*

Diaphragm 1 = 1st Floor: Weight = 10 x 4 = 40 kN
Diaphragm 2 = 2nd Floor: Weight = 16 x 4 = 64 kN
Diaphragm 3 = 3rd Floor: Weight = 23 x 4 = 92 kN
Verification Examples

V.06 Loading

<table>
<thead>
<tr>
<th>Diaphragm</th>
<th>Current Floor Weight (KN)</th>
<th>Above Floor Weight (KN)</th>
<th>Below Floor Weight (KN)</th>
<th>Current Floor Weight/Above Floor Weight</th>
<th>Current Floor Weight/Below Floor Weight</th>
<th>Status (FAIL if ratio &gt; 1.5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40</td>
<td>64</td>
<td>N.A.</td>
<td>0.625</td>
<td>N.A.</td>
<td>OK</td>
</tr>
<tr>
<td>2</td>
<td>64</td>
<td>92</td>
<td>40</td>
<td>0.696</td>
<td>1.6</td>
<td>FAIL</td>
</tr>
<tr>
<td>3</td>
<td>92</td>
<td>N.A.</td>
<td>64</td>
<td>1.4375</td>
<td>OK</td>
<td></td>
</tr>
</tbody>
</table>

Results

Table 458: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diaphragm 1</td>
<td>Ratio of 1st/2nd floor weights</td>
<td>0.625</td>
<td>0.625</td>
<td>none</td>
</tr>
<tr>
<td>Check</td>
<td>OK</td>
<td>OK</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Diaphragm 2</td>
<td>Ratio of 2nd/3rd floor weights</td>
<td>0.696</td>
<td>0.696</td>
<td>none</td>
</tr>
<tr>
<td>Check</td>
<td>OK</td>
<td>OK</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Ratio of 2nd/1st floor weights</td>
<td>1.6</td>
<td>1.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Check</td>
<td>Fail</td>
<td>Fail</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Diaphragm 3</td>
<td>Ratio of 3rd/2nd floor weights</td>
<td>1.438</td>
<td>1.437</td>
<td>negligible</td>
</tr>
<tr>
<td>Check</td>
<td>OK</td>
<td>OK</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2016\IS 1893 2016 Mass Irregularity.STD is typically installed with the program.
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;

DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL

MEMBER PROPERTY AMERICAN
1 TO 24 PRIS YD 0.5 ZD 0.5

CONSTANTS
MATERIAL CONCRETE ALL

SUPPORTS
1 4 5 8 FIXED

MEMBER CRACKED CODE IS1893 2016
1 3 6 8 TO 12 17 TO 20 REDUCTION RIY 0.7 RIZ 0.7
2 4 5 7 13 TO 16 21 TO 24 REDUCTION RIY 0.35 RIZ 0.35

DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1

JOINT LOAD
2 3 6 7 FY -10
9 TO 12 FY -16
13 TO 16 FY -23

END DEFINE REFERENCE LOADS

FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 5
DIA 2 TYPE RIG HEI 10
DIA 3 TYPE RIG HEI 15

CHECK IRREGULARITIES CODE IS1893 2016

DEFINE IS1893 2016 LOAD
ZONE 0.36 RF 5 I 1.2 SS 1 ST 5 DM 0.05
LOAD 1 LOADTYPE None TITLE STATIC_X
1893 LOAD X 1
LOAD 2 LOADTYPE None TITLE STATIC_Z
1893 LOAD Z 1
LOAD 3 LOADTYPE None TITLE STATIC_Y
1893 LOAD Y 1

PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH

**NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6 BASE SHEAR AND TIME PERIOD IN X

* UNITS - KN METE
* TIME PERIOD FOR X 1893 LOADING = 0.60374 SEC
 Verification Examples
 V.06 Loading

* SA/G PER 1893= 1.656, LOAD FACTOR= 1.000 *
* VB PER 1893= 0.0716 X 196.00= 14.02 KN *
* VB Act Based on Clause 7.2.1 = 14.02 KN *
* VB Min based on Clause 7.2.2 = 4.70 KN *
* *
*******************************************************************
**WARNING: BASE DIMENSION IS CALCULATED AT LOWEST SUPPORT
LEVEL OF THE STRUCTURE. IF THIS LEVEL IS NOT
PLINTH LEVEL USE PZ PARAMETER FOR TIME PERIOD
FOR Z 1893 LOADING.
***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
BASE SHEAR AND TIME PERIOD IN Z
*******************************************************************
* UNITS - KN METE *
* TIME PERIOD FOR Z 1893 LOADING = 0.60374 SEC *
* SA/G PER 1893 = 1.656, LOAD FACTOR = 1.000 *
* VB PER 1893= 0.0716 X 196.00= 14.02 KN *
* VB Act Based on Clause 7.2.1 = 14.02 KN *
* VB Min based on Clause 7.2.2 = 4.70 KN *
* *
*******************************************************************
STAAD SPACE -- PAGE NO. 5
***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
BASE SHEAR AND TIME PERIOD IN Y
*******************************************************************
* UNITS - KN METE *
* Calculation of SA/G for Y based on Clause 6.4.6 *
* SA/G PER 1893= 2.500, LOAD FACTOR= 1.000 *
* VB PER 1893= 0.0720 X 196.00= 14.11 KN *
* *
*******************************************************************
joint        lateral          torsional         load - 1
load (kn)    moment (kn -mete) factor - 1.000
-----        --------          ----------
2     FX    0.125    MY        0.000
3     FX    0.125    MY        0.000
6     FX    0.125    MY        0.000
7     FX    0.125    MY        0.000

TOTAL = 0.499 0.000 AT LEVEL 5.000 METE

VB PER 1893 = 14.025 KN
9     FX    0.799    MY        0.000
10    FX    0.799    MY        0.000
11    FX    0.799    MY        0.000
12    FX    0.799    MY        0.000

----------        ----------
TOTAL = 3.194 0.000 AT LEVEL 10.000 METE

VB PER 1893 = 14.025 KN
13    FX    2.583    MY        0.000
14    FX    2.583    MY        0.000
15    FX    2.583    MY        0.000
<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (KN)</th>
<th>Toroidal Load (KN)</th>
<th>Factor</th>
<th>Total</th>
<th>VB per 1893</th>
<th>Level (Meters)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>15,000</td>
</tr>
<tr>
<td>16</td>
<td>FX</td>
<td>MY</td>
<td>0.000</td>
<td>2.583</td>
<td>10.331</td>
<td>14.025</td>
</tr>
</tbody>
</table>

**Total:** 10.331 0.000

**VB per 1893:** 14.025 0.000

**Level:** 15.000

**Load:** 2

**Moment (KN-m):** 1.000

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (KN)</th>
<th>Toroidal Load (KN)</th>
<th>Factor</th>
<th>Total</th>
<th>VB per 1893</th>
<th>Level (Meters)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>5.000</td>
</tr>
<tr>
<td>9</td>
<td>FZ</td>
<td>MY</td>
<td>0.000</td>
<td>0.799</td>
<td>0.499</td>
<td>14.025</td>
</tr>
</tbody>
</table>

**Total:** 0.499 0.000

**VB per 1893:** 14.025 0.000

**Level:** 5.000

**Load:** 3

**Moment (KN-m):** 1.000

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (KN)</th>
<th>Toroidal Load (KN)</th>
<th>Factor</th>
<th>Total</th>
<th>VB per 1893</th>
<th>Level (Meters)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>10.000</td>
</tr>
<tr>
<td>13</td>
<td>FY</td>
<td>MY</td>
<td>0.000</td>
<td>2.599</td>
<td>3.194</td>
<td>14.025</td>
</tr>
</tbody>
</table>

**Total:** 3.194 0.000

**VB per 1893:** 14.025 0.000

**Level:** 10.000

**Load:** 3

**Moment (KN-m):** 1.000

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (KN)</th>
<th>Toroidal Load (KN)</th>
<th>Factor</th>
<th>Total</th>
<th>VB per 1893</th>
<th>Level (Meters)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>5.000</td>
</tr>
<tr>
<td>10</td>
<td>FY</td>
<td>MY</td>
<td>0.000</td>
<td>0.804</td>
<td>0.502</td>
<td>14.112</td>
</tr>
</tbody>
</table>

**Total:** 0.502 0.000

**VB per 1893:** 14.112 0.000

**Level:** 5.000

**Load:** 3

**Moment (KN-m):** 1.000

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (KN)</th>
<th>Toroidal Load (KN)</th>
<th>Factor</th>
<th>Total</th>
<th>VB per 1893</th>
<th>Level (Meters)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>10.000</td>
</tr>
<tr>
<td>16</td>
<td>FY</td>
<td>MY</td>
<td>0.000</td>
<td>2.599</td>
<td>3.214</td>
<td>14.112</td>
</tr>
</tbody>
</table>

**Total:** 3.214 0.000

**VB per 1893:** 14.112 0.000

**Level:** 10.000

**Load:** 3

**Moment (KN-m):** 1.000

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (KN)</th>
<th>Toroidal Load (KN)</th>
<th>Factor</th>
<th>Total</th>
<th>VB per 1893</th>
<th>Level (Meters)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>15.000</td>
</tr>
<tr>
<td>16</td>
<td>FY</td>
<td>MY</td>
<td>0.000</td>
<td>2.599</td>
<td>10.396</td>
<td>14.112</td>
</tr>
</tbody>
</table>

**Total:** 10.396 0.000

**VB per 1893:** 14.112 0.000

**Level:** 15.000

**Load:** 3

**Moment (KN-m):** 1.000

---

**Verification Examples**

**V.06 Loading**

---

**STAAD Space**

---

**Page 7**

---

**-Irregularity Checks**

---
--TORSION IRREGULARITY CHECKS
Torsion Irregularity Check
Ref: Table 5 (i) - Ratio Limit: 1.50

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Extreme Points of Dia in X</th>
<th>Extreme Points of Dia in Z</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Node</td>
<td>Disp. (mm)</td>
</tr>
<tr>
<td>1</td>
<td>7</td>
<td>0.24199</td>
</tr>
<tr>
<td>2</td>
<td>12</td>
<td>0.85035</td>
</tr>
<tr>
<td>3</td>
<td>16</td>
<td>1.62595</td>
</tr>
</tbody>
</table>

Diaphragm D X-max/min D Z-max/min Status

1 1.0000 1.0000 OK
2 1.0000 1.0000 OK
3 1.0000 1.0000 OK

--GEOMETRY IRREGULARITY CHECKS
Re-Entrant Corner Check.
(Ref: Table 5 (ii) - Ratio Limit: 0.15 )

***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.

--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Table 6 (ii) - Ratio Limit: 1.50

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Level (m)</th>
<th>Mass (kN)</th>
<th>Above (kN)</th>
<th>Below (kN)</th>
<th>Ratio Above</th>
<th>Ratio Below</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.000</td>
<td>40.000</td>
<td>64.000</td>
<td>Base</td>
<td>0.625</td>
<td>N/A</td>
<td>OK</td>
</tr>
<tr>
<td>2</td>
<td>10.000</td>
<td>64.000</td>
<td>92.000</td>
<td>40.000</td>
<td>0.696</td>
<td>1.600</td>
<td>FAIL</td>
</tr>
<tr>
<td>3</td>
<td>15.000</td>
<td>92.000</td>
<td>Top</td>
<td>64.000</td>
<td>N/A</td>
<td>1.437</td>
<td>OK</td>
</tr>
</tbody>
</table>

***NOTE: Linear dynamic analysis needs to carried out for Irregular Modes of Oscillation check.
***NOTE: Static Seismic Loads for relevant code needs to be defined with Zone 4 and 5 for Irregular Modes of Oscillation check.

55. PRINT ANALYSIS RESULTS
ANALYSIS RESULTS
STAAD SPACE -- PAGE NO.

8

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE
------------------------------------------
<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.1169</td>
<td>0.0013</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0013</td>
<td>0.1169</td>
<td>0.0003</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0003</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.1169</td>
<td>-0.0013</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0013</td>
<td>0.1169</td>
<td>0.0003</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0003</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>4</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.06 Loading

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>7</td>
<td>0.1169</td>
<td>0.0013</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
<td>0.0000</td>
</tr>
<tr>
<td>8</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>9</td>
<td>0.1169</td>
<td>0.0013</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>10</td>
<td>0.3039</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
<td>0.0000</td>
</tr>
<tr>
<td>11</td>
<td>0.3039</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>12</td>
<td>0.4558</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
</tr>
<tr>
<td>13</td>
<td>0.4558</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>14</td>
<td>0.4558</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>15</td>
<td>0.4558</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>16</td>
<td>0.4558</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

#### Support Reactions -UNIT KN METE

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-3.51</td>
<td>-13.66</td>
<td>0.00</td>
<td>0.00</td>
<td>13.19</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>-13.66</td>
<td>-3.51</td>
<td>-13.19</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-3.53</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**STAAD Space**

---

**STAAD Space**

---
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-3.51</td>
<td>13.66</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>13.19</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>-13.66</td>
<td>-3.51</td>
<td>-13.19</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-3.53</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-3.51</td>
<td>-13.66</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>13.19</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>13.66</td>
<td>-3.51</td>
<td>-13.19</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-3.53</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>-3.51</td>
<td>13.66</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>13.19</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>13.66</td>
<td>-3.51</td>
<td>-13.19</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-3.53</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**STAAD SPACE**

---

**Member End Forces**

**Structure Type = SPACE**

---

**All Units Are -- KN METE (Local)**
### Verification Examples

#### V.06 Loading

**STAAD Pro**

---

### Member Load JT Axial Shear-Y Shear-Z Torsion Mom-Y Mom-Z

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>7</td>
<td>10</td>
<td>13.66</td>
<td>3.51</td>
<td>0.00</td>
<td>4.34</td>
<td>13.19</td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>-13.66</td>
<td>3.51</td>
<td>-0.00</td>
<td>3.51</td>
<td>-4.34</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>7</td>
<td>-3.53</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>3.53</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>2</td>
<td>13.66</td>
<td>-0.00</td>
<td>3.51</td>
<td>4.34</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-13.66</td>
<td>0.00</td>
<td>-3.51</td>
<td>-0.00</td>
<td>-0.00</td>
<td>8.57</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>-3.40</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>3.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD SPACE**

---

### Member End Forces Structure Type = Space

**All Units Are -- KN Meter (Local)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>3</td>
<td>8.50</td>
<td>3.38</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>8.57</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>8.50</td>
<td>-3.38</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>6</td>
<td>-8.50</td>
<td>3.38</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>8.50</td>
<td>0.00</td>
<td>3.38</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>7</td>
<td>8.50</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>8.50</td>
<td>0.00</td>
<td>3.38</td>
<td>-0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>9</td>
<td>0.00</td>
<td>-5.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>0.00</td>
<td>-5.25</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Pro**

---

### User Manual

---

### Verification Examples

#### V.06 Loading

**STAAD Pro**

---

### Member Load JT Axial Shear-Y Shear-Z Torsion Mom-Y Mom-Z

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>1</td>
<td>9</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>9</td>
<td>0.00</td>
<td>-5.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>5.25</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>10</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>0.00</td>
<td>-5.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>5.25</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Space**

---

### Member End Forces Structure Type = Space

**All Units Are -- KN Meter (Local)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>1</td>
<td>9</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>9</td>
<td>0.00</td>
<td>-5.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>5.25</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>10</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>0.00</td>
<td>-5.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>5.25</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Pro**

---

### User Manual

---

### Verification Examples

#### V.06 Loading

**STAAD Pro**

---
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

#### MEMBER END FORCES

**STRUCTURE TYPE = SPACE**

**ALL UNITS ARE -- KN METE (LOCAL )**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>11</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.00</td>
<td>5.25</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-13.13</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**STAAD.Pro** 3529 User Manual
Related Links

- TR.28.2.2 Check Irregularities (on page 2531)

V. IS 1893 2016 Re entrant Corners

Verify reentrant corner (plan irregularity) check in accordance to IS 1893 2016 Part 1 specifications.

Details

A two story structure is modelled with diaphragms at the story levels.

Figure 418: Isometric View
Seismic parameters:

- Seismic Zone Factor, \( Z = 0.36 \)
- Importance Factor, \( I = 1.2 \)
- Response Reduction Factor, \( R = 5 \)
- Site soil conditions: hard soil (SS 1)
- Structure type: other (ST 5)

**Verification**

Both diaphragms have the same dimensions and are doubly-symmetric. The overall building width in either direction is 12 m and the potential reentrant corner sides are 4 m. Therefore, the ratio of the \( A/L = 4\ m / 12\ m = 0.333 > 0.15 \).

Thus, each of the four corners are reentrant corners per the IS 1893 2016 code.
Results

Table 459: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node 1</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Node 4</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Node 7</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Node 10</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Node 25</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Node 28</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Node 31</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Node 34</td>
<td>Reentrant</td>
<td>Reentrant</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\2016\IS 1893 2016 Reentrant Corners.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 07-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 3 0 0; 2 7 0 0; 3 7 0 4; 4 3 0 4; 5 3 0 -4; 6 -1 0 -4; 7 -1 0 0; 8 -5 0 0; 9 -5 0 4; 10 -1 0 4; 11 -1 0 8; 12 3 0 8; 13 3 -5 0; 14 7 -5 0; 15 7 -5 4; 16 3 -5 4; 17 3 -5 -4; 18 -1 -5 -4; 19 -1 -5 0; 20 -5 -5 0; 21 -5 -5 4; 22 -1 -5 4; 23 -1 -5 8; 24 3 -5 8; 25 3 5 0; 26 7 5 0; 27 7 5 4; 28 3 5 4; 29 3 5 -4; 30 -1 5 -4; 31 -1 5 0; 32 -5 5 0; 33 -5 5 4; 34 -1 5 4; 35 -1 5 8; 36 3 5 8;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 1 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10; 10 10 11; 11 11 12; 12 12 4; 13 1 13; 14 2 14; 15 3 15; 16 4 16; 17 5 17; 18 6 18; 19 7 19; 20 8 20; 21 9 21; 22 10 22; 23 11 23; 24 12 24; 25 1 25; 26 2 26; 27 3 27; 28 4 28; 29 5 29; 30 6 30; 31 7 31; 32 8 32; 33 9 33; 34 10 34; 35 11 35; 36 12 36; 37 25 26; 38 26 27; 39 27 28; 40 25 29; 41 29 30; 42 30 31; 43 31 32; 44 32 33; 45 33 34; 46 34 35; 47 35 36; 48 36 28;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 48 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
13 TO 24 FIXED
MEMBER CRACKED CODE IS1893 2016
13 TO 36 REDUCTION RY 0.7 RZ 0.7
1 TO 12 37 TO 48 REDUCTION RY 0.35 RZ 0.35
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 0
DIA 2 TYPE RIG HEI 5
CHECK IRREGULARITIES CODE IS1893 2016
DEFINE IS1893 2016 LOAD
ZONE 0.36 RF 5 I 1.2 SS 1 ST 5 DM 0.05
LOAD 1 LOADTYPE None TITLE STATIC_X
1893 LOAD X 1
LOAD 2 LOADTYPE None TITLE STATIC_Z
1893 LOAD Z 1
LOAD 3 LOADTYPE None TITLE STATIC_Y
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH

***NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
BASE SHEAR AND TIME PERIOD IN X

(units - KN METE
* TIME PERIOD FOR X 1893 LOADING = 0.25981 SEC *
* SA/G PER 1893= 2.500, LOAD FACTOR= 1.000 *
* VB PER 1893= 0.1080 X 1272.3= 137.41 KN *
* VB Act Based on Clause 7.2.1 = 137.41 KN *
* VB Min based on Clause 7.2.2 = 30.54 KN *

***NOTE: BASE DIMENSION IS CALCULATED AT LOWEST SUPPORT LEVEL OF THE STRUCTURE. IF THIS LEVEL IS NOT PLINTH LEVEL USE PZ PARAMETER FOR TIME PERIOD FOR Z 1893 LOADING.
***NOTE: Equivalent Static Analysis should preferably be performed for regular
structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6

BASE SHEAR AND TIME PERIOD IN Z

*********************************************************
* UNITS - KN    METE                                    *
* TIME PERIOD FOR Z 1893 LOADING =    0.25981 SEC       *
* SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
* VB PER 1893= 0.1000 X 1272.33= 137.41 KN             *
* VB Act Based on Clause 7.2.1 = 137.41 KN              *
* VB Min based on Clause 7.2.2 = 30.54 KN               *
*                                                       *
*********************************************************

STAAD SPACE -- PAGE NO. 5

***NOTE: Equivalent Static Analysis should preferably be performed for regular structures with approximate natural time period less than 0.4s and regular structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6

BASE SHEAR AND TIME PERIOD IN Y

*********************************************************
* UNITS - KN    METE                                    *
* Calculation of SA/G for Y based on Clause 6.4.6        *
* SA/G PER 1893= 2.500, LOAD FACTOR= 1.000              *
* VB PER 1893= 0.0720 X 1272.33= 91.61 KN               *
*                                                       *
*********************************************************

JOINT | LATERAL LOAD (KN) | TORSIONAL MOMENT (KN -METE) | LOAD - 1 FACTOR - 1.000
------ | ------------------ | ---------------------------- | ----------------------------
 1 | FX | 2.945 | MY | 0.000 |
 2 | FX | 2.945 | MY | 0.000 |
 3 | FX | 2.945 | MY | 0.000 |
 4 | FX | 2.945 | MY | 0.000 |
 5 | FX | 2.945 | MY | 0.000 |
 6 | FX | 2.945 | MY | 0.000 |
 7 | FX | 2.945 | MY | 0.000 |
 8 | FX | 2.945 | MY | 0.000 |
 9 | FX | 2.945 | MY | 0.000 |
10 | FX | 2.945 | MY | 0.000 |
11 | FX | 2.945 | MY | 0.000 |
12 | FX | 2.945 | MY | 0.000 |

TOTAL = 35.334 0.000 AT LEVEL 0.000 METE

VB PER 1893 = 137.411 KN

25 | FX | 8.506 | MY | 0.000 |
26 | FX | 8.506 | MY | 0.000 |
27 | FX | 8.506 | MY | 0.000 |
28 | FX | 8.506 | MY | 0.000 |
29 | FX | 8.506 | MY | 0.000 |
30 | FX | 8.506 | MY | 0.000 |
31 | FX | 8.506 | MY | 0.000 |
32 | FX | 8.506 | MY | 0.000 |
33 | FX | 8.506 | MY | 0.000 |
34 | FX | 8.506 | MY | 0.000 |
35 | FX | 8.506 | MY | 0.000 |
36 | FX | 8.506 | MY | 0.000 |

TOTAL = 102.077 0.000 AT LEVEL 5.000 METE
<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN)</th>
<th>TORSIONAL LOAD (KN)</th>
<th>LOAD - 2 FACTOR - 1.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 FZ</td>
<td>2.945</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>2 FZ</td>
<td>2.945</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

TOTAL = 35.334 | 0.000 AT LEVEL 0.000 METE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN)</th>
<th>TORSIONAL LOAD (KN)</th>
<th>LOAD - 3 FACTOR - 1.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>2 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>3 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>4 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>5 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>6 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>7 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>8 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>9 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>10 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>11 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>12 FY</td>
<td>1.963</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

TOTAL = 23.556 | 0.000 AT LEVEL 0.000 METE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN)</th>
<th>TORSIONAL LOAD (KN)</th>
<th>LOAD - 3 FACTOR - 1.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>26 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>27 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>28 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>29 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>30 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>31 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>32 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>33 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>34 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>35 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>36 FZ</td>
<td>8.506</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

TOTAL = 102.077 | 0.000 AT LEVEL 5.000 METE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN)</th>
<th>TORSIONAL LOAD (KN)</th>
<th>LOAD - 3 FACTOR - 1.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>26 FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>27 FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>28 FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

TOTAL = 23.556 | 0.000 AT LEVEL 0.000 METE
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Node</th>
<th>FY</th>
<th>Disp. (mm)</th>
<th>MY</th>
<th>Disp. (mm)</th>
<th>Disp. (mm)</th>
<th>Disp. (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>29</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>30</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>31</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>32</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>33</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>34</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>35</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>36</td>
<td>FY</td>
<td>5.671</td>
<td>MY</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

**TOTAL =** 68.051  0.000 AT LEVEL 5.000 METE

VB PER 1893 = 91.608 KN

************* END OF DATA FROM INTERNAL STORAGE *************

---

### IRREGULARITY CHECKS

**STAAD.PRO IRREGULARITIES CHECK - ( IS1893-2016 ) v1.0**

**TORSION IRREGULARITY CHECKS**

Torsion Irregularity Check

Ref: Table 5 (i)  - Ratio Limit: 1.50

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Extreme Points of Dia in X</th>
<th>Extreme Points of Dia in Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node</td>
<td>Disp. (mm)</td>
<td>Node</td>
</tr>
<tr>
<td>------</td>
<td>------------</td>
<td>------</td>
</tr>
<tr>
<td>1</td>
<td>1.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>2</td>
<td>1.0000</td>
<td>1.0000</td>
</tr>
</tbody>
</table>

**GEOMETRY IRREGULARITY CHECKS**

Re-Entrant Corner Check.

(Ref: Table 5 (ii)  - Ratio Limit: 0.15 )

<table>
<thead>
<tr>
<th>Node</th>
<th>Re-Entrant X-Proj (m)</th>
<th>X-Proj/Lx</th>
<th>Z-Proj (m)</th>
<th>Z-Proj/Lz</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>5</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>8</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>9</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>10</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>11</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>12</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>13</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>26</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>29</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>32</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>35</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
<td>0.3333</td>
<td>Re-Entrant</td>
</tr>
<tr>
<td>27</td>
<td>4.0000</td>
<td>0.3333</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
</tbody>
</table>

Diaphragm | Lx: (m) | Lz: (m) |
Verification Examples
V.06 Loading

--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Table 6 (ii) - Ratio Limit: 1.50

<table>
<thead>
<tr>
<th>Dia.</th>
<th>Level</th>
<th>Mass</th>
<th>Above</th>
<th>Below</th>
<th>Ratio Above</th>
<th>Ratio Below</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>636.163</td>
<td>459.451</td>
<td>Base</td>
<td>1.385</td>
<td>N/A</td>
<td>OK</td>
</tr>
<tr>
<td>2</td>
<td>5.000</td>
<td>459.451</td>
<td>Top</td>
<td>636.163</td>
<td>N/A</td>
<td>0.722</td>
<td>OK</td>
</tr>
</tbody>
</table>

***NOTE: Linear dynamic analysis needs to be carried out for Irregular Modes of Oscillation check.***

***NOTE: Static Seismic Loads for relevant code need to be defined with Zone 4 and 5 for Irregular Modes of Oscillation check.***

9. PRINT ANALYSIS RESULTS

**STAAD SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.3133</td>
<td>0.0027</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0006</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>-0.0027</td>
<td>0.3133</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.3133</td>
<td>-0.0028</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0006</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.3133</td>
<td>-0.0028</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0006</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>-0.0027</td>
<td>0.3133</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.3133</td>
<td>0.0027</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.3133</td>
<td>-0.0027</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0006</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.3133</td>
<td>0.0027</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>0.3133</td>
<td>-0.0027</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>0.3133</td>
<td>0.0028</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0.3133</td>
<td>0.0028</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>-0.0027</td>
<td>0.3133</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0.3133</td>
<td>-0.0027</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0006</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0.3133</td>
<td>0.0027</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>-0.0028</td>
<td>0.3133</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>0.3133</td>
<td>-0.0027</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0006</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>-0.0028</td>
<td>0.3133</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
</tr>
</tbody>
</table>
### JOINT DISPLACEMENT (CM  RADIANS)  STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>22</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>24</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>25</td>
<td>1</td>
<td>0.6965</td>
<td>0.0038</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>-0.0038</td>
<td>0.6965</td>
<td>0.0004</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>1</td>
<td>0.6965</td>
<td>-0.0039</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0038</td>
<td>0.6965</td>
<td>0.0004</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>1</td>
<td>0.6965</td>
<td>-0.0039</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0038</td>
<td>0.6965</td>
<td>0.0004</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>1</td>
<td>0.6965</td>
<td>0.0038</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0038</td>
<td>0.6965</td>
<td>0.0004</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>1</td>
<td>0.6965</td>
<td>-0.0038</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0039</td>
<td>0.6965</td>
<td>0.0004</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>1</td>
<td>0.6965</td>
<td>0.0038</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
</tr>
</tbody>
</table>

Verification Examples

V.06 Loading
### Verification Examples

#### V.06 Loading

**JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>31</td>
<td>1</td>
<td>0.6965</td>
<td>-0.0038</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>-0.0038</td>
<td>0.6965</td>
<td>0.0004</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

**SUPPORT REACTIONS -UNIT KN METE STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>1</td>
<td>-11.45</td>
<td>-29.84</td>
<td>-0.05</td>
<td>-0.09</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>2</td>
<td>-0.05</td>
<td>29.84</td>
<td>-11.45</td>
<td>-38.75</td>
<td>0.00</td>
<td>0.09</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>-11.45</td>
<td>30.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>2</td>
<td>-0.05</td>
<td>29.85</td>
<td>-11.45</td>
<td>-38.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>-11.45</td>
<td>30.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>2</td>
<td>0.05</td>
<td>29.85</td>
<td>-11.45</td>
<td>-38.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>-11.45</td>
<td>29.84</td>
<td>0.05</td>
<td>0.09</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>2</td>
<td>0.05</td>
<td>29.84</td>
<td>-11.45</td>
<td>-38.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>-11.45</td>
<td>29.85</td>
<td>0.05</td>
<td>-0.08</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>2</td>
<td>-0.00</td>
<td>30.52</td>
<td>-11.45</td>
<td>-38.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>-11.45</td>
<td>29.85</td>
<td>0.05</td>
<td>0.08</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>30.52</td>
<td>-11.45</td>
<td>-38.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>-11.45</td>
<td>29.84</td>
<td>0.05</td>
<td>0.09</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>2</td>
<td>0.05</td>
<td>29.84</td>
<td>-11.45</td>
<td>-38.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>-11.45</td>
<td>30.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>MEMBER</td>
<td>LOAD</td>
<td>JT</td>
<td>AXIAL</td>
<td>SHEAR-Y</td>
<td>SHEAR-Z</td>
<td>TORSION</td>
<td>MOM-Y</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td>----</td>
<td>-------</td>
<td>--------</td>
<td>--------</td>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>-18.29</td>
<td>0.00</td>
<td>-0.06</td>
<td>-0.00</td>
<td>-36.58</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>18.29</td>
<td>-0.00</td>
<td>0.06</td>
<td>-0.00</td>
<td>-36.58</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>-0.32</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.64</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.32</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.64</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>-0.32</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.64</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.32</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.64</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>18.29</td>
<td>0.00</td>
<td>0.06</td>
<td>-0.00</td>
<td>36.58</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>-18.29</td>
<td>0.00</td>
<td>0.11</td>
<td>0.00</td>
<td>-36.58</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>18.29</td>
<td>0.00</td>
<td>-0.11</td>
<td>0.00</td>
<td>-36.58</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>0.32</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.64</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>-0.32</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.64</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>-18.29</td>
<td>0.00</td>
<td>-0.06</td>
<td>0.00</td>
<td>36.58</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>36.58</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.00</td>
<td>0.32</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.64</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>-0.32</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0.00</td>
<td>-18.29</td>
<td>0.00</td>
<td>-0.06</td>
<td>0.00</td>
<td>-36.58</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>18.29</td>
<td>0.00</td>
<td>0.06</td>
<td>0.00</td>
<td>-36.58</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>0.00</td>
<td>-18.29</td>
<td>0.00</td>
<td>-0.06</td>
<td>0.00</td>
<td>-36.58</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>18.29</td>
<td>0.00</td>
<td>0.06</td>
<td>0.00</td>
<td>-36.58</td>
<td></td>
</tr>
<tr>
<td>MEMBER LOAD</td>
<td>JT</td>
<td>AXIAL</td>
<td>SHEAR-Y</td>
<td>SHEAR-Z</td>
<td>TORSION</td>
<td>MOM-Y</td>
<td>MOM-Z</td>
</tr>
<tr>
<td>-------------</td>
<td>----</td>
<td>-------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
<td>-------</td>
<td>-------</td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>0.00</td>
<td>-0.32</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.64</td>
</tr>
<tr>
<td>8</td>
<td>7</td>
<td>0.00</td>
<td>0.32</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.64</td>
</tr>
<tr>
<td>3</td>
<td>7</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>8</td>
<td>7</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>8</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>9</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>8</td>
<td>0.00</td>
<td>-18.29</td>
<td>-0.00</td>
<td>-0.11</td>
<td>0.00</td>
<td>-36.58</td>
</tr>
<tr>
<td>9</td>
<td>0.00</td>
<td>18.29</td>
<td>0.00</td>
<td>0.11</td>
<td>0.00</td>
<td>-36.58</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>8</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>10</td>
<td>2</td>
<td>30.52</td>
<td>11.45</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>18.51</td>
</tr>
<tr>
<td>14</td>
<td>-30.52</td>
<td>-11.45</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>14</td>
<td>-29.84</td>
<td>0.00</td>
<td>-11.45</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>11</td>
<td>-29.84</td>
<td>0.00</td>
<td>-11.45</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
</tr>
<tr>
<td>11</td>
<td>-7.63</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>Member</td>
<td>Load</td>
<td>JT</td>
<td>Axial</td>
<td>Shear Y</td>
<td>Shear Z</td>
<td>Torsion</td>
<td>MOM Y</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td>----</td>
<td>-------</td>
<td>--------</td>
<td>--------</td>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>3</td>
<td>30.52</td>
<td>11.45</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>15</td>
<td>-30.52</td>
<td>-11.45</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
<td>-0.17</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>29.85</td>
<td>-0.05</td>
<td>11.45</td>
<td>0.00</td>
<td>-18.51</td>
<td>-0.08</td>
</tr>
<tr>
<td>15</td>
<td>-29.85</td>
<td>0.05</td>
<td>-11.45</td>
<td>-0.00</td>
<td>-38.75</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>15</td>
<td>7.63</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>4</td>
<td>-29.84</td>
<td>11.45</td>
<td>-0.05</td>
<td>-0.00</td>
<td>0.17</td>
</tr>
<tr>
<td>16</td>
<td>29.84</td>
<td>-11.45</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
<td>-0.09</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>-29.84</td>
<td>0.05</td>
<td>11.45</td>
<td>-0.00</td>
<td>-38.75</td>
<td>-0.08</td>
</tr>
<tr>
<td>16</td>
<td>29.84</td>
<td>0.05</td>
<td>-11.45</td>
<td>-0.00</td>
<td>-38.75</td>
<td>-0.08</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>-7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>16</td>
<td>7.63</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>5</td>
<td>29.85</td>
<td>11.45</td>
<td>0.05</td>
<td>-0.00</td>
<td>0.17</td>
</tr>
<tr>
<td>17</td>
<td>-29.85</td>
<td>-11.45</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
<td>-0.08</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>-30.52</td>
<td>0.00</td>
<td>11.45</td>
<td>0.00</td>
<td>-18.51</td>
<td>0.00</td>
</tr>
<tr>
<td>17</td>
<td>30.52</td>
<td>0.00</td>
<td>-11.45</td>
<td>-0.00</td>
<td>-38.75</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>-7.63</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>17</td>
<td>7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>6</td>
<td>-29.85</td>
<td>11.45</td>
<td>-0.05</td>
<td>-0.00</td>
<td>0.17</td>
</tr>
<tr>
<td>18</td>
<td>29.85</td>
<td>-11.45</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
<td>-0.08</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>-30.52</td>
<td>0.00</td>
<td>11.45</td>
<td>0.00</td>
<td>-18.51</td>
<td>0.00</td>
</tr>
<tr>
<td>18</td>
<td>30.52</td>
<td>0.00</td>
<td>-11.45</td>
<td>-0.00</td>
<td>-38.75</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>-7.63</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>18</td>
<td>7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>7</td>
<td>29.84</td>
<td>11.45</td>
<td>-0.05</td>
<td>-0.00</td>
<td>0.17</td>
</tr>
<tr>
<td>19</td>
<td>-29.84</td>
<td>-11.45</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
<td>-0.08</td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>29.84</td>
<td>0.05</td>
<td>11.45</td>
<td>0.00</td>
<td>-18.51</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>-29.84</td>
<td>0.05</td>
<td>-11.45</td>
<td>-0.00</td>
<td>-38.75</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>7</td>
<td>-7.63</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>7.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>8</td>
<td>-30.52</td>
<td>11.45</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>30.52</td>
<td>-11.45</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38.75</td>
<td>0.00</td>
</tr>
</tbody>
</table>
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>27</td>
<td>1</td>
<td>3</td>
<td>12.23</td>
<td>8.51</td>
<td>0.03</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.05</td>
</tr>
<tr>
<td>27</td>
<td>2</td>
<td>3</td>
<td>11.88</td>
<td>-0.17</td>
<td>-8.51</td>
<td>0.00</td>
<td>-0.10</td>
<td>24.46</td>
</tr>
<tr>
<td>27</td>
<td>3</td>
<td>3</td>
<td>-5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>27</td>
<td>27</td>
<td>5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>28</td>
<td>4</td>
<td>1</td>
<td>-11.87</td>
<td>8.51</td>
<td>0.20</td>
<td>-0.00</td>
<td>-0.05</td>
<td>18.07</td>
</tr>
<tr>
<td>28</td>
<td>2</td>
<td>11.87</td>
<td>8.51</td>
<td>-0.20</td>
<td>-8.51</td>
<td>0.00</td>
<td>-0.57</td>
<td>24.46</td>
</tr>
<tr>
<td>28</td>
<td>2</td>
<td>11.88</td>
<td>0.17</td>
<td>8.51</td>
<td>0.00</td>
<td>24.46</td>
<td>0.47</td>
<td>0.47</td>
</tr>
<tr>
<td>28</td>
<td>3</td>
<td>-5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>28</td>
<td>28</td>
<td>5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>29</td>
<td>5</td>
<td>1</td>
<td>11.88</td>
<td>8.51</td>
<td>-0.17</td>
<td>-0.00</td>
<td>0.36</td>
<td>24.46</td>
</tr>
<tr>
<td>29</td>
<td>2</td>
<td>11.88</td>
<td>-0.17</td>
<td>8.51</td>
<td>0.00</td>
<td>0.47</td>
<td>24.46</td>
<td>0.05</td>
</tr>
<tr>
<td>29</td>
<td>2</td>
<td>-12.23</td>
<td>0.03</td>
<td>-8.51</td>
<td>0.00</td>
<td>18.07</td>
<td>0.05</td>
<td>18.08</td>
</tr>
<tr>
<td>29</td>
<td>3</td>
<td>-5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>29</td>
<td>29</td>
<td>5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>30</td>
<td>6</td>
<td>1</td>
<td>-11.88</td>
<td>8.51</td>
<td>0.17</td>
<td>-0.00</td>
<td>-0.36</td>
<td>18.08</td>
</tr>
<tr>
<td>30</td>
<td>30</td>
<td>11.88</td>
<td>-0.17</td>
<td>8.51</td>
<td>0.00</td>
<td>-0.47</td>
<td>24.46</td>
<td>0.05</td>
</tr>
<tr>
<td>30</td>
<td>2</td>
<td>-12.23</td>
<td>0.03</td>
<td>-8.51</td>
<td>0.00</td>
<td>18.07</td>
<td>-0.05</td>
<td>-0.05</td>
</tr>
<tr>
<td>30</td>
<td>3</td>
<td>-5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>30</td>
<td>30</td>
<td>5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>31</td>
<td>7</td>
<td>1</td>
<td>11.87</td>
<td>8.51</td>
<td>0.20</td>
<td>-0.00</td>
<td>-0.41</td>
<td>18.07</td>
</tr>
<tr>
<td>31</td>
<td>31</td>
<td>11.87</td>
<td>-0.20</td>
<td>-8.51</td>
<td>0.00</td>
<td>-0.57</td>
<td>24.46</td>
<td>0.41</td>
</tr>
<tr>
<td>31</td>
<td>2</td>
<td>11.87</td>
<td>0.20</td>
<td>8.51</td>
<td>0.00</td>
<td>24.46</td>
<td>0.57</td>
<td>0.57</td>
</tr>
<tr>
<td>31</td>
<td>3</td>
<td>-5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>31</td>
<td>31</td>
<td>5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>32</td>
<td>8</td>
<td>1</td>
<td>-12.23</td>
<td>8.51</td>
<td>0.03</td>
<td>-0.00</td>
<td>-0.05</td>
<td>18.07</td>
</tr>
<tr>
<td>32</td>
<td>32</td>
<td>12.23</td>
<td>-8.51</td>
<td>-0.03</td>
<td>0.00</td>
<td>-0.10</td>
<td>24.46</td>
<td>0.41</td>
</tr>
</tbody>
</table>
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>8</td>
<td>-11.88</td>
<td>-0.17</td>
<td>-8.51</td>
<td>0.00</td>
<td>18.08</td>
<td>-0.36</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>-5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>1</td>
<td>-12.23</td>
<td>8.51</td>
<td>-0.03</td>
<td>-0.00</td>
<td>0.05</td>
<td>18.07</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>12.23</td>
<td>-8.51</td>
<td>0.03</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>24.46</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD SPACE** -- Page No.

---

### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>9</td>
<td>11.88</td>
<td>0.17</td>
<td>-8.51</td>
<td>0.00</td>
<td>18.08</td>
<td>0.36</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>-5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>1</td>
<td>9</td>
<td>-5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>10</td>
<td>11.87</td>
<td>8.51</td>
<td>-0.20</td>
<td>0.00</td>
<td>0.57</td>
<td>18.07</td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>10</td>
<td>-11.87</td>
<td>-8.51</td>
<td>0.20</td>
<td>0.00</td>
<td>24.46</td>
<td>0.57</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>-5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>11</td>
<td>5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>1</td>
<td>11</td>
<td>-11.88</td>
<td>8.51</td>
<td>-0.17</td>
<td>0.00</td>
<td>18.08</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>9</td>
<td>-11.88</td>
<td>-11.88</td>
<td>-0.17</td>
<td>0.00</td>
<td>24.46</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>11</td>
<td>-5.67</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>12</td>
<td>-5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>12</td>
<td>5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>1</td>
<td>11</td>
<td>-11.88</td>
<td>8.51</td>
<td>-0.17</td>
<td>0.00</td>
<td>24.46</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>10</td>
<td>-11.88</td>
<td>-8.51</td>
<td>0.17</td>
<td>0.00</td>
<td>24.46</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>-5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>11</td>
<td>5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>37</td>
<td>1</td>
<td>25</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>-0.13</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>25</td>
<td>0.00</td>
<td>12.23</td>
<td>0.00</td>
<td>0.13</td>
<td>-0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>26</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>26</td>
<td>0.00</td>
<td>12.23</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>38</td>
<td>1</td>
<td>25</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>25</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>26</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>26</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>27</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>1</td>
<td>27</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.13</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>27</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>28</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.36</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD SPACE** -- Page No.
Verification Examples
V.06 Loading

<p>| | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>1</td>
<td>25</td>
<td>0.00</td>
<td>0.36</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.70</td>
</tr>
<tr>
<td>29</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.36</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.73</td>
</tr>
<tr>
<td>2</td>
<td>25</td>
<td>0.00</td>
<td>12.23</td>
<td>-0.00</td>
<td>-0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>24.46</td>
</tr>
<tr>
<td>29</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>24.46</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>25</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>41</td>
<td>1</td>
<td>29</td>
<td>0.00</td>
<td>12.23</td>
<td>0.00</td>
<td>0.26</td>
<td>-0.00</td>
<td>24.47</td>
</tr>
<tr>
<td>30</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>-0.26</td>
<td>-0.00</td>
<td>-0.00</td>
<td>24.47</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.03</td>
</tr>
<tr>
<td>30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.03</td>
</tr>
<tr>
<td>3</td>
<td>29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>42</td>
<td>1</td>
<td>30</td>
<td>0.00</td>
<td>0.36</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.73</td>
</tr>
<tr>
<td>31</td>
<td>0.00</td>
<td>-0.36</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.70</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>30</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>-24.46</td>
</tr>
<tr>
<td>31</td>
<td>0.00</td>
<td>12.23</td>
<td>0.00</td>
<td>-0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>-24.46</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>31</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>43</td>
<td>1</td>
<td>31</td>
<td>0.00</td>
<td>12.23</td>
<td>0.00</td>
<td>-0.13</td>
<td>-0.00</td>
<td>24.46</td>
</tr>
<tr>
<td>32</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>0.13</td>
<td>-0.00</td>
<td>-0.00</td>
<td>24.46</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>31</td>
<td>0.00</td>
<td>-0.36</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.70</td>
</tr>
<tr>
<td>32</td>
<td>0.00</td>
<td>0.36</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.73</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>31</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>44</td>
<td>1</td>
<td>32</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.03</td>
</tr>
<tr>
<td>33</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.03</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>32</td>
<td>0.00</td>
<td>-12.23</td>
<td>-0.00</td>
<td>-0.26</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-24.47</td>
</tr>
<tr>
<td>33</td>
<td>0.00</td>
<td>12.23</td>
<td>0.00</td>
<td>0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>-24.47</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>45</td>
<td>1</td>
<td>33</td>
<td>0.00</td>
<td>-12.23</td>
<td>0.00</td>
<td>0.13</td>
<td>-0.00</td>
<td>-24.46</td>
</tr>
<tr>
<td>34</td>
<td>0.00</td>
<td>12.23</td>
<td>-0.00</td>
<td>-0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>-24.46</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>33</td>
<td>0.00</td>
<td>-0.36</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.73</td>
</tr>
<tr>
<td>34</td>
<td>0.00</td>
<td>0.36</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.70</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>46</td>
<td>1</td>
<td>34</td>
<td>0.00</td>
<td>-0.36</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.70</td>
</tr>
<tr>
<td>35</td>
<td>0.00</td>
<td>0.36</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.73</td>
<td></td>
</tr>
</tbody>
</table>
V. Moving Load

V. Moving Load Generator

To validate the effect of wheel load assembly on bridge girders simulated by a series of equivalent concentrated loads for a particular wheel position, generated by Moving Load generation option.

Details

A bridge decking system of 10m span is simulated by three longitudinal girder of size 0.3m×0.6m (Concrete) at a spacing of 2m and six transverse girders of size 0.3m×0.45m (Concrete) at a spacing of 2m. Longitudinal girders are pinned support at both the ends.

Vehicle definition – 10 kN and 16 kN per wheel in the Front and Rear Axle respectively. The vehicle width is 1.2m and the axle distance is 1.5m. Starting position is from the end of the deck, aligning the rear axle along the last transverse girder. The direction of movement along the longitudinal girder with an incremental distance of 0.5m.

Validation

Among the generated wheel loads, for the 9th position of the wheel assembly, the structure is analyzed for that generated load. The structure is again analyzed with the static load system equivalent to the generated wheel load and observed the difference in results if any.

![Figure 420: The position of the generated load case 9](image-url)
Note: The postprocessing diagrams in STAAD.Pro show two overlapping 7 kN axle reactions on member 14.

The model uses the moving load generator facility and a static load equivalent to the generated Load Case 9 (i.e., 9th position of the wheel assembly). Both support reactions and joint displacements are compared.

Comparison

<table>
<thead>
<tr>
<th>Parameter</th>
<th>STAAD.Pro</th>
<th>Reference</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Support reaction at joint 7 (kN)</td>
<td>11.37</td>
<td>11.37</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Vertical displacement at joint 10 (cm)</td>
<td>0.3025</td>
<td>0.3025</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Moving Load\Moving Load Generator.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
J OINT COORDINATES
1 0 0 0; 2 10 0 0; 3 2 0 0; 4 4 0 0; 5 6 0 0; 6 8 0 0; 7 0 0 2; 8 10 0 2;
9 2 0 2; 10 4 0 2; 11 6 0 2; 12 8 0 2; 13 0 0 4; 14 10 0 4; 15 2 0 4; 16 4 0 4;
17 6 0 4; 18 8 0 4;
MEMBER INCIDENCES
1 1 3; 2 3 4; 3 4 5; 4 5 6; 5 6 2; 6 1 7; 7 2 8; 8 3 9; 9 4 10; 10 5 11;
11 6 12; 12 7 9; 13 9 10; 14 10 11; 15 11 12; 16 12 8; 17 7 13; 18 8 14;
19 9 15; 20 10 16; 21 11 17; 22 12 18; 23 13 15; 24 15 16; 25 16 17; 26 17 18;
27 18 14;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 12 TO 16 23 TO 27 PRIS YD 0.6 ZD 0.3
6 TO 11 17 TO 22 PRIS YD 0.45 ZD 0.3
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 2 7 8 13 14 PINNED
DEFINE MOVING LOAD
TYPE 1 LOAD 16 10
DIST 1.5 WID 1.2
LOAD GENERATION 18
TYPE 1 0 0 2.6 XINC 0.5
LOAD 19 LOADTYPE Live TITLE STATIC POS9
MEMBER LOAD
9 CON GY -16 1.4
20 CON GY -16 0.6
14 CON GY -14 1.5
3 25 CON GY -3 1.5
PERFORM ANALYSIS PRINT LOAD DATA
LOAD LIST 9 19
PRINT SUPPORT REACTION ALL
PRINT JOINT DISPLACEMENTS ALL
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>LOADING</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER LOAD - UNIT KN</td>
<td>METE</td>
</tr>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>17</td>
<td>-16.0000</td>
</tr>
<tr>
<td>6</td>
<td>-16.0000</td>
</tr>
<tr>
<td>23</td>
<td>-3.0000</td>
</tr>
<tr>
<td>12</td>
<td>-7.0000</td>
</tr>
<tr>
<td>12</td>
<td>-7.0000</td>
</tr>
<tr>
<td>1</td>
<td>-3.0000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LOADING</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER LOAD - UNIT KN</td>
<td>METE</td>
</tr>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>23</td>
<td>-4.8000</td>
</tr>
<tr>
<td>12</td>
<td>-11.2000</td>
</tr>
<tr>
<td>12</td>
<td>-11.2000</td>
</tr>
<tr>
<td>1</td>
<td>-4.8000</td>
</tr>
<tr>
<td>19</td>
<td>-10.0000</td>
</tr>
<tr>
<td>8</td>
<td>-10.0000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LOADING</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER LOAD - UNIT KN</td>
<td>METE</td>
</tr>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>23</td>
<td>-4.8000</td>
</tr>
<tr>
<td>12</td>
<td>-11.2000</td>
</tr>
<tr>
<td>12</td>
<td>-11.2000</td>
</tr>
<tr>
<td>1</td>
<td>-4.8000</td>
</tr>
<tr>
<td>24</td>
<td>-3.0000</td>
</tr>
<tr>
<td>13</td>
<td>-7.0000</td>
</tr>
<tr>
<td>13</td>
<td>-7.0000</td>
</tr>
<tr>
<td>2</td>
<td>-3.0000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LOADING</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>STAAD SPACE</td>
<td>-- PAGE NO.</td>
</tr>
<tr>
<td>MEMBER LOAD - UNIT KN  METE</td>
<td></td>
</tr>
<tr>
<td>---------------------------</td>
<td></td>
</tr>
<tr>
<td>MEMBER UDL L1 L2 CON L LIN1 LIN2</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-4.8000 GY 1.50</td>
</tr>
<tr>
<td>12</td>
<td>-11.2000 GY 1.50</td>
</tr>
<tr>
<td>12</td>
<td>-11.2000 GY 1.50</td>
</tr>
<tr>
<td>1</td>
<td>-4.8000 GY 1.50</td>
</tr>
<tr>
<td>24</td>
<td>-3.0000 GY 1.00</td>
</tr>
<tr>
<td>13</td>
<td>-7.0000 GY 1.00</td>
</tr>
<tr>
<td>13</td>
<td>-7.0000 GY 1.00</td>
</tr>
<tr>
<td>2</td>
<td>-3.0000 GY 1.00</td>
</tr>
</tbody>
</table>

LOADING 5

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN METE</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER UDL L1 L2 CON L LIN1 LIN2</td>
</tr>
<tr>
<td>19</td>
</tr>
<tr>
<td>8</td>
</tr>
<tr>
<td>24</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

LOADING 6

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN METE</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER UDL L1 L2 CON L LIN1 LIN2</td>
</tr>
<tr>
<td>24</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>20</td>
</tr>
<tr>
<td>9</td>
</tr>
</tbody>
</table>

LOADING 7

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN METE</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER UDL L1 L2 CON L LIN1 LIN2</td>
</tr>
<tr>
<td>24</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>25</td>
</tr>
<tr>
<td>14</td>
</tr>
<tr>
<td>14</td>
</tr>
<tr>
<td>3</td>
</tr>
</tbody>
</table>

LOADING 8

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN METE</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER UDL L1 L2 CON L LIN1 LIN2</td>
</tr>
<tr>
<td>24</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>25</td>
</tr>
<tr>
<td>14</td>
</tr>
<tr>
<td>14</td>
</tr>
<tr>
<td>3</td>
</tr>
</tbody>
</table>

LOADING 9

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN METE</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER UDL L1 L2 CON L LIN1 LIN2</td>
</tr>
<tr>
<td>24</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>25</td>
</tr>
<tr>
<td>14</td>
</tr>
<tr>
<td>14</td>
</tr>
<tr>
<td>3</td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.06 Loading

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>Con</th>
<th>L</th>
<th>Lin1</th>
<th>Lin2</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-16.0000 GY</td>
<td>0.60</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-16.0000 GY</td>
<td>1.40</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
</tbody>
</table>

**Loading 10**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>Con</th>
<th>L</th>
<th>Lin1</th>
<th>Lin2</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-4.8000 GY</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-11.2000 GY</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-11.2000 GY</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-4.8000 GY</td>
<td>0.50</td>
<td></td>
</tr>
</tbody>
</table>

STAAD SPACE

---

**Page No.**

6

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>Con</th>
<th>L</th>
<th>Lin1</th>
<th>Lin2</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-10.0000 GY</td>
<td>0.60</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-10.0000 GY</td>
<td>1.40</td>
<td></td>
</tr>
</tbody>
</table>

**Loading 11**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>Con</th>
<th>L</th>
<th>Lin1</th>
<th>Lin2</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-4.8000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-11.2000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-11.2000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-4.8000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>0.50</td>
<td></td>
</tr>
</tbody>
</table>

**Loading 12**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>Con</th>
<th>L</th>
<th>Lin1</th>
<th>Lin2</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-4.8000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-11.2000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-11.2000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-4.8000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>1.00</td>
<td></td>
</tr>
</tbody>
</table>

**Loading 13**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>Con</th>
<th>L</th>
<th>Lin1</th>
<th>Lin2</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-16.0000 GY</td>
<td>0.60</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-16.0000 GY</td>
<td>1.40</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-7.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-3.0000 GY</td>
<td>1.50</td>
<td></td>
</tr>
</tbody>
</table>

STAAD SPACE

---

**Page No.**

7

**Loading 14**

---
### Verification Examples

#### V.06 Loading

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN</th>
<th>META</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>26</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>15</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>15</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>4</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>22</td>
<td>-10.0000 GY</td>
</tr>
<tr>
<td>11</td>
<td>-10.0000 GY</td>
</tr>
<tr>
<td>LOADING</td>
<td>15</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN</th>
<th>META</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>26</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>15</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>15</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>4</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>27</td>
<td>-3.0000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-7.0000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-7.0000 GY</td>
</tr>
<tr>
<td>5</td>
<td>-3.0000 GY</td>
</tr>
<tr>
<td>LOADING</td>
<td>16</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN</th>
<th>META</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>26</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>15</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>15</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>4</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>27</td>
<td>-3.0000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-7.0000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-7.0000 GY</td>
</tr>
<tr>
<td>5</td>
<td>-3.0000 GY</td>
</tr>
<tr>
<td>LOADING</td>
<td>17</td>
</tr>
</tbody>
</table>

#### STAAD SPACE

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN</th>
<th>META</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>27</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>5</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>18</td>
<td>-10.0000 GY</td>
</tr>
<tr>
<td>7</td>
<td>-10.0000 GY</td>
</tr>
<tr>
<td>LOADING</td>
<td>19</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MEMBER LOAD - UNIT KN</th>
<th>META</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER</td>
<td>UDL</td>
</tr>
<tr>
<td>27</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>16</td>
<td>-11.2000 GY</td>
</tr>
<tr>
<td>5</td>
<td>-4.8000 GY</td>
</tr>
<tr>
<td>18</td>
<td>-10.0000 GY</td>
</tr>
<tr>
<td>7</td>
<td>-10.0000 GY</td>
</tr>
</tbody>
</table>
### SUPPORT REACTIONS -UNIT KN METE STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9</td>
<td>0.00</td>
<td>8.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.00</td>
<td>8.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>9</td>
<td>0.00</td>
<td>7.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.00</td>
<td>7.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td>9</td>
<td>0.00</td>
<td>11.37</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.00</td>
<td>11.37</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>8</td>
<td>9</td>
<td>0.00</td>
<td>8.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.00</td>
<td>8.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>13</td>
<td>9</td>
<td>0.00</td>
<td>8.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.00</td>
<td>8.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>14</td>
<td>9</td>
<td>0.00</td>
<td>7.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.00</td>
<td>7.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0009</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0009</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0009</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0009</td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>0.0000</td>
<td>-0.1712</td>
<td>0.0000</td>
<td>0.0001</td>
<td>0.0000</td>
<td>-0.0007</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.1712</td>
<td>0.0000</td>
<td>0.0001</td>
<td>0.0000</td>
<td>-0.0007</td>
</tr>
<tr>
<td>4</td>
<td>9</td>
<td>0.0000</td>
<td>-0.2777</td>
<td>0.0000</td>
<td>0.0002</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.2777</td>
<td>0.0000</td>
<td>0.0002</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>5</td>
<td>9</td>
<td>0.0000</td>
<td>-0.2731</td>
<td>0.0000</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0003</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.2731</td>
<td>0.0000</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0003</td>
</tr>
<tr>
<td>6</td>
<td>9</td>
<td>0.0000</td>
<td>-0.1645</td>
<td>0.0000</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0007</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.1645</td>
<td>0.0000</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0007</td>
</tr>
<tr>
<td>7</td>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0010</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0010</td>
</tr>
<tr>
<td>8</td>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0009</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0009</td>
</tr>
<tr>
<td>9</td>
<td>9</td>
<td>0.0000</td>
<td>-0.1827</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0008</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.1827</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0008</td>
</tr>
<tr>
<td>10</td>
<td>9</td>
<td>0.0000</td>
<td>-0.3025</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.3025</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>11</td>
<td>9</td>
<td>0.0000</td>
<td>-0.2915</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0004</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.2915</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0004</td>
</tr>
<tr>
<td>12</td>
<td>9</td>
<td>0.0000</td>
<td>-0.1709</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0008</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>0.0000</td>
<td>-0.1709</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0008</td>
</tr>
</tbody>
</table>


### Verification Examples

#### V.06 Loading

<table>
<thead>
<tr>
<th>13</th>
<th>9</th>
<th>0.0000</th>
<th>0.0000</th>
<th>0.0000</th>
<th>-0.0000</th>
<th>0.0000</th>
<th>-0.0009</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td></td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0009</td>
</tr>
<tr>
<td>14</td>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0009</td>
</tr>
<tr>
<td>19</td>
<td></td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0009</td>
</tr>
<tr>
<td>15</td>
<td>9</td>
<td>0.0000</td>
<td>-0.1712</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>-0.0007</td>
</tr>
<tr>
<td>19</td>
<td></td>
<td>0.0000</td>
<td>-0.1712</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>-0.0007</td>
</tr>
<tr>
<td>16</td>
<td>9</td>
<td>0.0000</td>
<td>-0.2777</td>
<td>0.0000</td>
<td>-0.0002</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>19</td>
<td></td>
<td>0.0000</td>
<td>-0.2777</td>
<td>0.0000</td>
<td>-0.0002</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>17</td>
<td>9</td>
<td>0.0000</td>
<td>-0.2731</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0003</td>
</tr>
<tr>
<td>19</td>
<td></td>
<td>0.0000</td>
<td>-0.2731</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0003</td>
</tr>
<tr>
<td>18</td>
<td>9</td>
<td>0.0000</td>
<td>-0.1645</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0007</td>
</tr>
<tr>
<td>19</td>
<td></td>
<td>0.0000</td>
<td>-0.1645</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0007</td>
</tr>
</tbody>
</table>

**Related Links**
- *M. Moving Loads* (on page 853)
- *EX. US-12 Moving Load Generation on a Bridge Deck* (on page 4454)
- *EX. UK-12 Moving Load Generation on a Bridge Deck* (on page 4735)
- *TR.31.1 Definition of Moving Load System* (on page 2541)

#### V. NRC

**V. NRC 2005**

**V. NRC 2005 Response Spectrum**


**Reference**

1. Example 4.7, [http://nptel.ac.in/courses/105101004/4](http://nptel.ac.in/courses/105101004/4) (p.36)

**Related Links**
- *TR.32.10.1.2 Response Spectrum Specification per NRC 2005* (on page 2694)
- *Response Spectra tab* (on page 3021)

**Details**

Example 4.7(1) is modeled in STAAD.Pro.

The generated model is subjected to a Response Spectrum Load along global X direction. Time-Acceleration pairs for the spectrum data can be seen in supporting excel file *NRC_Calculation.xlsx* within worksheet named RS Values. The direction factor along global X direction is assumed as 0.5. The base shear reported by STAAD.Pro is verified against hand calculation.

**Validation**

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)
### Table 461: Natural Frequencies/Time Period

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (cyc/sec)</th>
<th>Time Period (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

### Table 462: Mode Shapes

<table>
<thead>
<tr>
<th>Mode</th>
<th>Story Level/ Joint Numbers</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>-0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### Table 463: Mode Participation Factor Calculation

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th></th>
<th>Mode 2</th>
<th></th>
<th>Mode 3</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>φ</td>
<td>Wi x φ</td>
<td>Wi x φ</td>
<td></td>
<td>φ</td>
<td>Wi x φ</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>49.035</td>
<td>49.035</td>
<td>-1</td>
<td>-49.035</td>
<td>49.035</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Verification Examples*

**V.06 Loading**

STAAD.Pro

3554

User Manual
## Table 464: Horizontal Acceleration Spectrum Value Calculation

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$\phi$</td>
<td>$Wi \times \phi$</td>
<td>$Wi \times \phi$</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86603</td>
<td>84.932</td>
<td>73.553</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>49.035</td>
<td>24.518</td>
</tr>
<tr>
<td>Summation</td>
<td>245.175</td>
<td>183.00</td>
<td>147.11</td>
<td></td>
</tr>
<tr>
<td>Modal Weight Mk x g</td>
<td></td>
<td>227.7</td>
<td></td>
<td>16.345</td>
</tr>
<tr>
<td>Modal Weight Participation (In %)</td>
<td>92.85</td>
<td></td>
<td>6.667</td>
<td></td>
</tr>
<tr>
<td>Mode Participation Factor $\prod \sum Wi \times \phi / \sum Wi \times \phi^2$</td>
<td>1.244</td>
<td></td>
<td>0.3333</td>
<td></td>
</tr>
</tbody>
</table>

### Table:

<table>
<thead>
<tr>
<th>Time Period T</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.30014</td>
<td>0.10986</td>
<td>0.08042</td>
<td></td>
</tr>
</tbody>
</table>

**Spectrum Data**: Obtain from NRC_Calculation.xlsx within worksheet named RS Values

**Formula**

- Linearly Interpolate acceleration values between 0.3 sec & 0.36 sec to obtain acceleration value for 0.30014 sec.
- Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.10986 sec.
- Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.08042 sec.

**Sa**

- 1.50
- 1.40
- 1.10

**Horizontal Seismic Coefficient Cs**

- Cs = (Sa x Direction Factor) = Sa x 0.5
- 249998725
- 1.25
- 1.10
Table 465: Base Shear Calculation and its distribution across height

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Vi (Qi)</td>
<td>Vi (Qi)</td>
<td>Vi (Qi)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Vi (Qi)</td>
<td>Vi (Qi)</td>
<td>Vi (Qi)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Vi (Qi)</td>
<td>Vi (Qi)</td>
<td>Vi (Qi)</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>76.25</td>
<td>-1</td>
<td>-19.60</td>
</tr>
<tr>
<td></td>
<td></td>
<td>76.25</td>
<td>-19.60</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>76.25</td>
<td>-19.60</td>
<td>4.608</td>
</tr>
<tr>
<td></td>
<td></td>
<td>76.25</td>
<td>-19.60</td>
<td>4.608</td>
</tr>
<tr>
<td></td>
<td></td>
<td>76.25</td>
<td>-19.60</td>
<td>78.86</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86603</td>
<td>132.1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>208.3</td>
<td>0</td>
<td>-19.60</td>
</tr>
<tr>
<td></td>
<td></td>
<td>208.3</td>
<td>0</td>
<td>-19.60</td>
</tr>
<tr>
<td></td>
<td></td>
<td>208.3</td>
<td>0</td>
<td>-0.86603</td>
</tr>
<tr>
<td></td>
<td></td>
<td>208.3</td>
<td>0</td>
<td>-0.86603</td>
</tr>
<tr>
<td></td>
<td></td>
<td>208.3</td>
<td>0</td>
<td>-7.981</td>
</tr>
<tr>
<td></td>
<td></td>
<td>208.3</td>
<td>0</td>
<td>-3.373</td>
</tr>
<tr>
<td></td>
<td></td>
<td>208.3</td>
<td>0</td>
<td>209.27</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>76.25</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>284.6</td>
<td>39.21</td>
<td>19.60</td>
</tr>
<tr>
<td></td>
<td></td>
<td>284.6</td>
<td>39.21</td>
<td>1.235</td>
</tr>
<tr>
<td></td>
<td></td>
<td>284.6</td>
<td>39.21</td>
<td>1.235</td>
</tr>
<tr>
<td></td>
<td></td>
<td>284.6</td>
<td>39.21</td>
<td>285.25</td>
</tr>
</tbody>
</table>

Comparison

Table 466: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>STAAD.Pro</th>
<th>Reference</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base shear (kN)</td>
<td>285.25</td>
<td>285.25</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\V06 Loading\NRC\2005\NRC 2005 Response Spectrum.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISOMATIC COLUMN
TABLE 2
UNIT METER KN
PRISMATIC BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None TITLE RS_X
SPECTRUM SRSS NRC 2005 X 0.5 ACC DAMP 0.05 LIN
0 9.80665; 0.06 18.6326; 0.12 24.5166; 0.18 24.5166; 0.24 24.5166; 0.3 24.5166;
0.36 24.5166; 0.42 23.3492; 0.48 20.4305; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6203; 0.78 12.5726; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.6146; 1.08 9.0083; 1.14 8.6023; 1.2 8.1721;
1.26 7.7836; 1.32 7.4292; 1.38 7.1062; 1.44 6.8108; 1.5 6.5777;
1.56 6.2863; 1.62 6.0534; 1.68 5.8372; 1.74 5.6361; 1.8 5.4481;
1.86 5.2724; 1.92 5.1076; 1.98 4.9528; 2.04 4.8071; 2.1 4.6698;
2.16 4.5401; 2.22 4.4174; 2.28 4.3011; 2.34 4.1908; 2.4 4.0861;
2.46 3.9864; 2.52 3.8915; 2.58 3.8018; 2.64 3.7146; 2.7 3.6329;
2.76 3.5531; 2.82 3.4775; 2.88 3.4059; 2.94 3.3356; 3.0 3.2689; 3.06 3.2047;
3.12 3.1431; 3.18 3.0838; 3.24 3.0267; 3.3 2.9717; 3.36 2.9186;
3.42 2.8674; 3.48 2.818; 3.54 2.7702; 3.6 2.7240; 3.66 2.6794; 3.72 2.6362;
3.78 2.5943; 3.84 2.5538; 3.9 2.5145; 3.96 2.4764; 4.02 2.4516;
4.08 2.4516; 4.14 2.4516; 4.2 2.4516; 4.26 2.4516; 4.32 2.4516;
4.38 2.4516; 4.44 2.4516; 4.5 2.4516; 4.56 2.4516; 4.62 2.4516;
4.68 2.4516; 4.74 2.4516; 4.8 2.4516; 4.86 2.4516; 4.92 2.4516;
4.98 2.4516; 5.04 2.4516; 5.1 2.4516; 5.16 2.4516; 5.22 2.4516;
5.28 2.4516; 5.34 2.4516; 5.4 2.4516; 5.46 2.4516; 5.52 2.4516;
### STAAD Output

#### CALCULATED FREQUENCIES FOR LOAD CASE 1

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (CYCLES/SEC)</th>
<th>PERIOD (SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

**RESPONSE SPECTRUM LOAD 1**

**RESPONSE LOAD CASE 1**

### MODAL WEIGHT (MODAL MASS TIMES g) IN KN

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.276565E+02</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>2</td>
<td>1.634500E+01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471050E+02</td>
</tr>
<tr>
<td>3</td>
<td>1.173519E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>1.471051E+02</td>
</tr>
</tbody>
</table>

**SRSS MODAL COMBINATION METHOD USED.**

**DYNAMIC WEIGHT X Y Z** 2.451750E+02 0.000000E+00 0.000000E+00 KN

**MISSING WEIGHT X Y Z** -3.177132E-06 0.000000E+00 0.000000E+00 KN

**MODAL WEIGHT X Y Z** 2.451750E+02 0.000000E+00 0.000000E+00 KN

**MODE** | **ACCELERATION-G** | **DAMPING** |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.50000</td>
<td>0.05000</td>
</tr>
<tr>
<td>2</td>
<td>2.39857</td>
<td>0.05000</td>
</tr>
<tr>
<td>3</td>
<td>2.10421</td>
<td>0.05000</td>
</tr>
</tbody>
</table>

**STAAD SPACE** -- PAGE NO. 8

### MODAL BASE ACTIONS

#### FORCES IN KN LENGTH IN METE

<table>
<thead>
<tr>
<th>MODE</th>
<th>PERIOD</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.300</td>
<td>284.57</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-1707.42</td>
<td>19.60</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.080</td>
<td>1.23</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### PARTICIPATION FACTORS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>92.85</td>
<td>0.00</td>
<td>0.00</td>
<td>92.85</td>
<td>0.00</td>
<td>284.57</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>6.67</td>
<td>0.00</td>
<td>0.00</td>
<td>99.521</td>
<td>0.00</td>
<td>19.60</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.48</td>
<td>0.00</td>
<td>0.00</td>
<td>100.000</td>
<td>0.00</td>
<td>1.23</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>
V. NRC 2005 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static force method in NBCC/NRC 2005.

Problem

Structure to be modelled:

- height = 3× 5m = 15m
- plan dimension X = 5m
- plan dimension Z = 5m

Input parameters:

- Seismic acceleration values Sa(0.2) = 0.28, Sa(0.5) = 0.17, Sa(1.0) = 0.11, and Sa(2) = 0.063 (SA1 0.28, SA2 0.17, SA3 0.11, SA4 0.063)
- Importance factor = 1.3 (IE 1.3)
- Site class C (SCL 3)
- Higher mode factor X = 1 (MVX 1)
- Higher mode factor Z = 1 (MVZ 1)
- Numerical reduction coefficient for base overturning X = 1 (JX 1)
- Numerical reduction coefficient for base overturning Z = 1 (JZ 1)
- Ductility-related force modification factor X = 5 (RDX 5)
- Ductility-related force modification factor Z = 5 (RDZ 5)
- Overstrength-related force modification factor X = 5 (ROX 1.5)
- Overstrength-related force modification factor Z = 5 (ROZ 1.5)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 6 7 9 To 16. Seismic Load as per NBCC/NRC 2005 specifications are generated along horizontal direction global X and global Z. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

\[ T_{a_X} = 0.075 \times (hn)^{0.75} \] (for concrete moment frames) = 0.075 × 15^{0.75} = 0.571649342 sec

\[ T_{a_Z} = 0.075 \times (hn)^{0.75} \] (for concrete moment frames) = 0.075 × 15^{0.75} = 0.571649342 sec

\[ T_{c_X} \] (Rayleigh method from output file) = 0.467, \( T_{c_Z} \) (Rayleigh method from output file) = 0.467

\[ 1.5 \times T_{a_X} = 0.857474013 > T_{c_X} \] so \( T_{a_{used}} = T_{c_X} = 0.467 \)

\[ 1.5 \times T_{a_Z} = 0.857474013 > T_{c_Z} \] so \( T_{a_{used}} = T_{c_Z} = 0.467 \)

\( F_a = 1 \) (For Site Class = C and Sa(0.2) = 0.28)

\( F_v = 1 \) (For Site Class = C and Sa(1.0) = 0.11)

\[ S(0.2) = F_a \times Sa(0.2) = 1 \times 0.28 = 0.28 \]
\[ S(0.5) = \text{smaller of } F_v \times S_a(0.5) \text{ and } F_a \times S_a(0.2) = \text{smaller of } 1 \times 0.17 \text{ and } 1 \times 0.28 = 0.17 \]
\[ S(T_{\text{used}})_x = S(T_{\text{used}})_z = 0.17 + \frac{(0.28-0.17)}{(0.5-0.2)} \times (0.5-0.467) = 0.1821 \]
\[ V_x = S(T_{\text{used}})_x \times MV_x \times I \times W \div (RDX \times ROX) = 0.1821 \times 1 \times 1.3 \times (200 \times 3) \div 5 \times 1.5 = 18.9384 \text{ kN} \]
Min Base Shear = \[ S(2.0) \times MV_x \times I \times W \div (RDX \times ROX) = 0.063 \times 1 \times 1.3 \times (200 \times 3) \div 5 \times 1.5 = 6.552 \text{ kN} \]
Max Base Shear = \[ 2/3 \times S(0.2) \times I \times W \div (RDX \times ROX) = 2/3 \times 0.28 \times 1.3 \times (200 \times 3) \div 5 \times 1.5 = 19.413 \text{ kN} \]
\[ V_x > \text{Min Base Shear} \text{ & } V_x < \text{Max Base Shear}, \quad V_x = 18.9384 \text{ kN} \]

\[ V_z = S(T_{\text{used}})_z \times MV_z \times I \times W \div (RDZ \times ROZ) = 0.1821 \times 1 \times 1.3 \times (200 \times 3) \div 5 \times 1.5 = 18.9384 \text{ kN} \]
Min Base Shear = \[ S(2.0) \times MV_z \times I \times W \div (RDZ \times ROZ) = 0.063 \times 1 \times 1.3 \times (200 \times 3) \div 5 \times 1.5 = 6.552 \text{ kN} \]
Max Base Shear = \[ 2/3 \times S(0.2) \times I \times W \div (RDZ \times ROZ) = 2/3 \times 0.28 \times 1.3 \times (200 \times 3) \div 5 \times 1.5 = 19.413 \text{ kN} \]
\[ V_z > \text{Min Base Shear} \text{ & } V_z < \text{Max Base Shear}, \quad V_z = 18.9384 \text{ kN} \]

Since fundamental lateral period < 0.7 sec \( F_t = 0 \), calculated Base Shear is distributed along the height as under:
\[ F_x = (V - 0) \times (W_x \times h_x \div \sum W_i \times h_i) = (V) \times (W_x \times h_x \div \sum W_i \times h_i) \]

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Wi</th>
<th>hi</th>
<th>Wi × hi</th>
<th>((Wi \times hi)\div\sum(Wi \times hi))</th>
<th>Fx</th>
<th>Fz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>200</td>
<td>15</td>
<td>3,000</td>
<td>0.5</td>
<td>9.469</td>
<td>9.469</td>
</tr>
<tr>
<td>2nd</td>
<td>200</td>
<td>10</td>
<td>2,000</td>
<td>0.333</td>
<td>6.313</td>
<td>6.313</td>
</tr>
<tr>
<td>1st</td>
<td>200</td>
<td>5</td>
<td>1,000</td>
<td>1.667</td>
<td>3.156</td>
<td>3.156</td>
</tr>
<tr>
<td>Summation</td>
<td>600</td>
<td></td>
<td>6,000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>V</td>
<td></td>
<td></td>
<td>18.938</td>
<td></td>
<td>18.938</td>
<td></td>
</tr>
</tbody>
</table>

**Comparison**

Table 467: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X, 1st</td>
<td>3.156</td>
<td>3.159</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Z, 1st</td>
<td>3.156</td>
<td>3.159</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>X, 2nd</td>
<td>6.313</td>
<td>6.318</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Z, 2nd</td>
<td>6.313</td>
<td>6.318</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>X, roof</td>
<td>9.469</td>
<td>9.477</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Z, roof</td>
<td>9.469</td>
<td>9.477</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Base shear (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vx</td>
<td>18.938</td>
<td>18.955</td>
<td>Negligible</td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.06 Loading

#### Table

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vz</td>
<td>18.938</td>
<td>18.955</td>
<td>Negligible</td>
<td></td>
</tr>
</tbody>
</table>

#### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\NRC\2005\NRC 2005 Static Seismic.STD is typically installed with the program.

**STAAD SPACE**

**START JOB INFORMATION**

**ENGINEER DATE** 25-Aug-18

**END JOB INFORMATION**

**INPUT WIDTH** 79

**UNIT** METER KN

**JOINT COORDINATES**
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5; 9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5; 16 5 15 5;

**MEMBER INCIDENCES**
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10; 11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14; 19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;

**DEFINE MATERIAL START**

**ISOTROPIC CONCRETE**
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579

**END DEFINE MATERIAL**

**MEMBER PROPERTY AMERICAN**
1 TO 24 PRIS YD 0.5 ZD 0.5

**CONSTANTS**
MATERIAL CONCRETE ALL

**SUPPORTS**
1 4 5 8 FIXED

*DEFINE NRC 2010 LOAD
*SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1 MVZ 1 RDX 5 RDZ 5 -
*ROX 1.5 ROZ 1.5 STX 1 STZ 1 MD 1

**DEFINE NRC 2005 LOAD**
SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 IE 1.3 SCLASS 3 MVX 1 MVZ 1 JX 1 JZ 1 -
RDX 5 RDZ 5 ROX 1.5 ROZ 1.5

**JOINT WEIGHT**
2 3 6 7 9 TO 16 WEIGHT 50
LOAD 1 LOADTYPE None TITLE STATIC_X
NRC LOAD X 1
LOAD 2 LOADTYPE None TITLE STATIC_Z
NRC LOAD Z 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
SECTION 0 0.25 0.5 0.75 1 ALL
STAAD Output

*******************************************************************************************
*                                        *
* EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2005 ALONG X *
*  CT =  0.073 Ta =  0.557 SEC. Tc =  0.467  *
* SEC.                                     *
*  T USED =  0.467 SEC. DESIGN SPECTRAL ACCELERATION = 0.182258  *
* EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC RESPONSE)  *
*                                                  = 0.032 X  600.000 =  18.955 *
* DESIGN BASE SHEAR =  1.000 X               *
*                                                =  18.955 *
*                                                *
*                                                *
*******************************************************************************************

*******************************************************************************************
*                                        *
* EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2005 ALONG Z *
*  CT =  0.073 Ta =  0.557 SEC. Tc =  0.467  *
* SEC.                                     *
*  T USED =  0.467 SEC. DESIGN SPECTRAL ACCELERATION = 0.182258  *
* EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC RESPONSE)  *
*                                                  = 0.032 X  600.000 =  18.955 *
* DESIGN BASE SHEAR =  1.000 X               *
*                                                =  18.955 *
*                                                *
*                                                *
*******************************************************************************************

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KN)</th>
<th>TORSIONAL MOMENT (KN-METE)</th>
<th>LOAD FACTOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>FX 0.790</td>
<td>MY 0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>3</td>
<td>FX 0.790</td>
<td>MY 0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>6</td>
<td>FX 0.790</td>
<td>MY 0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>7</td>
<td>FX 0.790</td>
<td>MY 0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>
### Verification Examples

**V.06 Loading**

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kN)</th>
<th>Torsional Moment (kN·m)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.790</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>3</td>
<td>0.790</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>6</td>
<td>0.790</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>7</td>
<td>0.790</td>
<td>0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Total</th>
<th>3.159</th>
<th>0.000</th>
<th>5.000 METE</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kN)</th>
<th>Torsional Moment (kN·m)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>14</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>15</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>16</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Total</th>
<th>6.318</th>
<th>0.000</th>
<th>10.000 METE</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kN)</th>
<th>Torsional Moment (kN·m)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>10</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>11</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>12</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Total</th>
<th>3.159</th>
<th>0.000</th>
<th>5.000 METE</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kN)</th>
<th>Torsional Moment (kN·m)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>14</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>15</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>16</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Total</th>
<th>6.318</th>
<th>0.000</th>
<th>10.000 METE</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kN)</th>
<th>Torsional Moment (kN·m)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>10</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>11</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>12</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Total</th>
<th>3.159</th>
<th>0.000</th>
<th>5.000 METE</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kN)</th>
<th>Torsional Moment (kN·m)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>14</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>15</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>16</td>
<td>2.369</td>
<td>0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Total</th>
<th>6.318</th>
<th>0.000</th>
<th>10.000 METE</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kN)</th>
<th>Torsional Moment (kN·m)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>10</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>11</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
<tr>
<td>12</td>
<td>1.580</td>
<td>0.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Total</th>
<th>3.159</th>
<th>0.000</th>
<th>5.000 METE</th>
</tr>
</thead>
</table>

### Related Links
- [TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1](on page 2552)
- **V. NRC 2010**

### V. NRC 2010 Response Spectrum


### Reference

1. Example 4.7, [http://nptel.ac.in/courses/105101004/4](p.36) (p.36)

### Related Links
- [TR.32.10.1.3 Response Spectrum Specification per NRC 2010](on page 2702)
- [Response Spectra tab](on page 3021)
Details

Example 4.7(1) is modeled in STAAD.Pro.

The generated model is subjected to a Response Spectrum Load along global X direction. Time-Acceleration pairs for the spectrum data can be seen in supporting excel file NRC_Calculation.xlsx within worksheet named RS Values. The direction factor along global X direction is assumed as 0.5. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 468: Natural Frequencies/Time Period

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (cyc/sec)</th>
<th>Time Period (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

Table 469: Mode Shapes

<table>
<thead>
<tr>
<th>Mode</th>
<th>Story Level/Join Numbers</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rot</th>
<th>Y-Rot</th>
<th>Z-Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>0.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>1st Floor/ Joints 2 &amp; 3</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>Roof/Joints 7 &amp; 8</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2nd Floor/ Joints 5 &amp; 6</td>
<td>-0.86603</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
### Table 470: Mode Participation Factor Calculation

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$\phi$</td>
<td>$Wi x \phi$</td>
<td>$Wi x \phi^2$</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>1</td>
<td>49.035</td>
<td>49.035</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86603</td>
<td>84.932</td>
<td>73.553</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>49.035</td>
<td>24.518</td>
</tr>
<tr>
<td>Summation</td>
<td>245.175</td>
<td>183.00</td>
<td>147.11</td>
<td></td>
</tr>
<tr>
<td>Modal Weight Mk x g</td>
<td></td>
<td>227.7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Modal Weight Participation (In %)</td>
<td></td>
<td>92.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mode Participation Factor $\prod \Sigma Wi x \phi / \Sigma Wi x \phi^2$</td>
<td></td>
<td>1.244</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Table 471: Horizontal Acceleration Spectrum Value Calculation

<table>
<thead>
<tr>
<th>Time Period $T$</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.30014</td>
<td>0.10986</td>
<td>0.08042</td>
<td></td>
</tr>
</tbody>
</table>

**Spectrum Data**
- Obtain from NRC_Calculation.xlsx within worksheet named **RS Values**

**Formula**
- Linearly Interpolate acceleration values between 0.3 sec & 0.36 sec to obtain acceleration value for 0.30014 sec.
- Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.10986 sec.
- Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.08042 sec.
<table>
<thead>
<tr>
<th>Story Level</th>
<th>Weight Wi (KN)</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>( Qi (Cs x \phi x \prod x Wi) )</td>
<td>( Vi (\sum Qi) )</td>
<td>( Qi (Cs x \phi x \prod x Wi) )</td>
</tr>
<tr>
<td>Roof</td>
<td>49.035</td>
<td>76.25</td>
<td>76.25</td>
<td>-1</td>
</tr>
<tr>
<td>2nd Floor</td>
<td>98.07</td>
<td>0.86603</td>
<td>132.1</td>
<td>208.3</td>
</tr>
<tr>
<td>1st Floor</td>
<td>98.07</td>
<td>0.5</td>
<td>76.25</td>
<td>284.6</td>
</tr>
</tbody>
</table>

Table 472: Base Shear Calculation and its distribution across height

Comparison

Table 473: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>STAAD.Pro</th>
<th>Reference</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base shear (kN)</td>
<td>285.25</td>
<td>285.25</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\NRC\2010\NRC 2010 Response Spectrum.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC COLUUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLUUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None TITLE RS X
SPECTRUM SRSS NRC 2010 X 0.5 ACC DAMP 0.05 LIN
0 9.80665; 0.06 18.6326; 0.12 24.5166; 0.18 24.5166; 0.24 24.5166; 0.3
24.5166;
0.36 24.5166; 0.42 23.3492; 0.48 20.4305; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6203; 0.78 12.5726; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61436; 1.08 9.08023; 1.14 8.60233; 1.2 8.17221;
1.26 7.78306; 1.32 7.42928; 1.38 7.10627; 1.44 6.81018; 1.5 6.53777;
1.56 6.28632; 1.62 6.05349; 1.68 5.83729; 1.74 5.63601; 1.8 5.44814;
1.86 5.2724; 1.92 5.10763; 1.98 4.95286; 2.04 4.80718; 2.1 4.66984;
### STAAD Output

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency(Cycles/Sec)</th>
<th>Period(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.332</td>
<td>0.30014</td>
</tr>
<tr>
<td>2</td>
<td>9.103</td>
<td>0.10986</td>
</tr>
<tr>
<td>3</td>
<td>12.435</td>
<td>0.08042</td>
</tr>
</tbody>
</table>

**Response Spectrum Load**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Modal Weight (Modal Mass Times g) in KN</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Generalized</td>
</tr>
<tr>
<td>1</td>
<td>2.2765E+02 0.00000E+00 0.00000E+00 1.4710E+02</td>
</tr>
<tr>
<td>2</td>
<td>1.6345E+01 0.00000E+00 0.00000E+00 1.4710E+02</td>
</tr>
<tr>
<td>3</td>
<td>1.1735E+00 0.00000E+00 0.00000E+00 1.4710E+02</td>
</tr>
</tbody>
</table>

**SRSS**

- Dynamic Weight X Y Z: 2.4517E+02 0.00000E+00 0.00000E+00 KN
- Missing Weight X Y Z: -3.1771E-06 0.00000E+00 0.00000E+00 KN

**Modal Weight X Y Z:**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Acceleration-G</th>
<th>Damping</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.50000</td>
<td>0.05000</td>
</tr>
<tr>
<td>2</td>
<td>2.39857</td>
<td>0.05000</td>
</tr>
<tr>
<td>3</td>
<td>2.10421</td>
<td>0.05000</td>
</tr>
</tbody>
</table>

### Modal Base Actions

<table>
<thead>
<tr>
<th>Mode</th>
<th>Period</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.300</td>
<td>284.57</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-1707.42</td>
<td>0.110</td>
<td>19.60</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>58.81</td>
<td>1.23</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**About the Origin**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Period</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.300</td>
<td>284.57</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-1707.42</td>
<td>0.110</td>
<td>19.60</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>58.81</td>
<td>1.23</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>
PARTICIPATION FACTORS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
<th>BASE SHEAR IN KN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>92.85</td>
<td>0.00</td>
<td>0.00</td>
<td>92.855</td>
<td>0.000</td>
<td>0.000</td>
<td>284.57</td>
</tr>
<tr>
<td>2</td>
<td>6.67</td>
<td>0.00</td>
<td>0.00</td>
<td>99.521</td>
<td>0.000</td>
<td>0.000</td>
<td>19.60</td>
</tr>
<tr>
<td>3</td>
<td>0.48</td>
<td>0.00</td>
<td>0.00</td>
<td>100.000</td>
<td>0.000</td>
<td>0.000</td>
<td>1.23</td>
</tr>
</tbody>
</table>

-----------------------------
TOTAL SRSS SHEAR  285.25     0.00
TOTAL 10PCT SHEAR  285.25     0.00
TOTAL ABS SHEAR    305.41     0.00

V. NRC 2010 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static force method in NBCC/NRC 2010.

Problem

Structure to be modelled:

- height = 3× 5m = 15m
- plan dimension X = 5m
- plan dimension Z = 5m

Input parameters:

- Seismic acceleration values Sa(0.2) = 0.28, Sa(0.5) = 0.17, Sa(1.0) = 0.11, and Sa(2) = 0.063 (SA1 0.28, SA2 0.17, SA3 0.11, SA4 0.063)
- Importance factor = 1.3 (I 1.3)
- Site class C (SCL 3)
- Higher mode factor X = 1 (MVX 1)
- Higher mode factor Z = 1 (MVZ 1)
- Ductility-related force modification factor X = 5 (RDX 5)
- Ductility-related force modification factor Z = 5 (RDZ 5)
- Overstrength-related force modification factor X = 5 (ROX 1.5)
- Overstrength-related force modification factor Z = 5 (ROZ 1.5)
- Moment resisting frames in both X and Z directions (STX 1, STZ 1)
- Provisions to be checked for member strength (MD 1)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 6 7 9 To 16. Seismic Load as per NBCC/NRC 2010 specifications are generated along horizontal direction global X and global Z. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.
Ta = 0.075×(hn)$^{0.75}$ (for concrete moment frames) = 0.075 × 15$^{0.75}$ = 0.57 sec
Ta = 0.075×(hn)$^{0.75}$ (for concrete moment frames) = 0.075 × 15$^{0.75}$ = 0.57 sec
Tc (Rayleigh method from output file) = 0.467, Tc (Rayleigh method from output file) = 0.467
1.5 × Ta = 0.86 > Tc so Ta = Tc = 0.467
1.5 × Ta = 0.86 > Tc so Ta = Tc = 0.467

Fa = 1 (For Site Class = C and Sa(0.2) = 0.28)
Fv = 1 (For Site Class = C and Sa(1.0) = 0.11)

S(0.2) = Fa × Sa(0.2) = 1 × 0.28 = 0.28
S(0.5) = smaller of Fv × Sa(0.5) and Fa × Sa(0.2) = smaller of 1 x 0.17 and 1 × 0.28 = 0.17
S(Ta) = S(Ta) = 0.17 + ((0.28-0.17)/(0.5-0.2) × (0.5-0.467)) = 0.1821

Vx = S(Ta) × MVX × I × W / RDX × ROX = 0.1821 × 1 × 1.3 × (200x3) / 5 × 1.5 = 18.9384 kN
Min Base Shear = S(2.0) × MVX × I × W / (RDX × ROX) = 0.063 × 1 x 1.3 × (200x3) / 5 × 1.5 = 6.552 kN
Max Base Shear = 2/3 × S(0.2) × I × W / (RDX × ROX) = 2/3 × 0.28 x 1.3 × (200x3) / 5 × 1.5 = 19.413 kN

Vx > Min Base Shear & Vx < Max Base Shear, Vx = 18.9384 kN

Vz = S(Ta) × MVZ × I × W / RDZ × ROZ = 0.1821 × 1 × 1.3 × (200x3) / 5 × 1.5 = 18.9384 kN
Min Base Shear = S(2.0) × MVZ × I × W / (RDZ × ROZ) = 0.063 × 1 x 1.3 × (200x3) / 5 × 1.5 = 6.552 kN
Max Base Shear = 2/3 × S(0.2) × I × W / (RDZ × ROZ) = 2/3 × 0.28 x 1.3 × (200x3) / 5 × 1.5 = 19.413 kN

Vz > Min Base Shear & Vz < Max Base Shear, Vz = 18.9384 kN

Since fundamental lateral period < 0.7 sec Ft = 0, calculated Base Shear is distributed along the height as under:

Fx = (V - 0) × (Wx × hx / ∑ Wi × hi) = (V) × (Wx × hx / ∑ Wi × hi)

<table>
<thead>
<tr>
<th>Story Level</th>
<th>Wi</th>
<th>hi</th>
<th>Wi × hi</th>
<th>(Wi × hi)/∑(Wi × hi)</th>
<th>Fx</th>
<th>Fz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roof</td>
<td>200</td>
<td>15</td>
<td>3,000</td>
<td>0.5</td>
<td>9.469</td>
<td>9.469</td>
</tr>
<tr>
<td>2nd</td>
<td>200</td>
<td>10</td>
<td>2,000</td>
<td>0.333</td>
<td>6.313</td>
<td>6.313</td>
</tr>
<tr>
<td>1st</td>
<td>200</td>
<td>5</td>
<td>1,000</td>
<td>1.667</td>
<td>3.156</td>
<td>3.156</td>
</tr>
<tr>
<td>Summation</td>
<td>600</td>
<td></td>
<td>6,000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>V</td>
<td></td>
<td></td>
<td>18.938</td>
<td></td>
<td>18.938</td>
<td></td>
</tr>
</tbody>
</table>
Comparison

Table 474: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Story shear (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X, 1st</td>
<td>3.156</td>
<td>3.159</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Z, 1st</td>
<td>3.156</td>
<td>3.159</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>X, 2nd</td>
<td>6.313</td>
<td>6.318</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Z, 2nd</td>
<td>6.313</td>
<td>6.318</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>X, roof</td>
<td>9.469</td>
<td>9.477</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Z, roof</td>
<td>9.469</td>
<td>9.477</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Base shear (kN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vx</td>
<td>18.938</td>
<td>18.955</td>
<td>Negligible</td>
<td></td>
</tr>
<tr>
<td>Vz</td>
<td>18.938</td>
<td>18.955</td>
<td>Negligible</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\06 Loading\NRC\2010\NRC 2010 Static Seismic.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 24 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
DEFINE NRC 2010 LOAD
SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1 MVZ 1 RDX 5 RDZ 5 ROX -
1.5 ROZ 1.5 STX 1 STZ 1 MD 1
JOINT WEIGHT
2 3 6 7 9 To 16 WEIGHT 50
LOAD 1 LOADTYPE None TITLE STATIC_X
NRC LOAD X 1
LOAD 2 LOADTYPE None TITLE STATIC_Z
NRC LOAD Z 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
SECTION 0 0.25 0.5 0.75 1 ALL
PRINT MEMBER SECTION FORCES ALL
FINISH

STAAD Output

*****************************************************************************
*                                                                           *
* EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2010 ALONG  *
X *                                                                           *
* CT = 0.075 Ta = 0.572 SEC. Tc = 0.467                                    *
SEC.                                                                         *
* T USED = 0.467 SEC. DESIGN SPECTRAL ACCELERATION = 0.182258              *
* EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC                                     *
RESPONSE)                                                                    *
* = 0.032 X 600.000 = 18.955                                               *
KN                                                                           *
* DESIGN BASE SHEAR = 1.000 X                                               *
18.955                                                                       *
KN                                                                            *
*                                                                           *
*****************************************************************************

*****************************************************************************
*                                                                           *
* EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2010 ALONG  *
Z *                                                                           *
* CT = 0.075 Ta = 0.572 SEC. Tc = 0.467                                    *
SEC.                                                                         *
* T USED = 0.467 SEC. DESIGN SPECTRAL ACCELERATION = 0.182258              *
* EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC                                     *
RESPONSE)                                                                    *
* = 0.032 X 600.000 = 18.955                                               *
KN                                                                           *
* DESIGN BASE SHEAR = 1.000 X                                               *

**************************************************************************************************
Verification Examples

V.06 Loading

Related Links

- **TR.31.2.4 Canadian Seismic Code (NRC) - 2010** (on page 2556)
V. Wind Load

V. ASCE 7-10 Wind Load Generation

To validate STAAD.Pro calculated equivalent joint loads for a closed structure subjected to Wind Loading. Wind Intensity is calculated as per ASCE 07 – 2010.

Details

When closed type structures are subjected to wind load, panels or closed surfaces are generated by the program based on the members in the ranges specified and their end joints. Wind load, in such way, is converted to equivalent joint loads and applied on the nodes of that closed surface. The area within each closed surface is determined and the share of this area (influence area) for each node in the list is calculated. Wind intensities have been calculated as per ASCE 07 – 2010. The detailed calculation of determination of equivalent joint loads is performed here in this document.

A two-story, single-bay frame is subjected to wind loading from +X direction. Assumptions made while calculating the wind intensities are tabulated in the Validation section of this document.

Verification

Calculation of Wind Intensity

Assumptions made in determination of wind intensities as per ASCE 07 – 2010

i. Analysis Procedure – Directional Procedure for MWFRS of enclosed buildings, Chapter 27

ii. Type of roof – flat

iii. Building Classification Category = Category II (Table 1.5-1)

iv. Basic Wind Speed = 110 mph

v. Exposure Category = Exposure B (Clause 26.7.3)

vi. Wind speed-up over Hills or Escarpment is not considered

vii. Building Height = 20 ft

viii. Building length along the direction of Wind (L) = 10 ft

ix. Building length normal the direction of Wind (B) = 20 ft

x. Natural Frequency of the building = 2 Hz (Hence, as per clause 26.2, this is a rigid building)

xi. Building Damping Ratio = 0.05

xii. Enclosure Classification = Enclosed Building (Clause 26.10)

xiii. Directionality Factor Kd = 0.85 (Table 26.6-1, for MWFRS)

xiv. Topographic Factor, Kzt = 1 (Clause 26.8.2)

xv. Building Wall to generate Wind Load on – Windward side

xvi. Gust Effect Factor, G = 0.85 (Clause 26.9.1)

xvii. External pressure Coefficient Cp = 0.85 (Windward wall pressure coefficient, as per Table 27.4-1)

xviii. Internal pressure coefficient GCpi = -0.18 (For Enclosed Buildings, Table 26.11-1)

Hence, the velocity pressure exposure coefficients (Kz) are calculated as per the formulae given in Table 27.3-1:

From Table 26.9.1, For Exposure B, α = 7 and zg = 1200 ft
As per clause 27.3.2, velocity pressure $q_z = 0.00256x K_z x K_{zt} x K_d x V^2$ (lb/ft$^2$)

Since Mean roof height $h = $ Height of building = 20 ft

$$q_h = 0.00256x (K_z)_{h=20\ ft} x K_{zt} x K_d x V^2 = 16.42847 \ lb/ft^2$$

Design wind pressure, $p$ is calculated as per clause 27.4.1 as -

$$p = q x G x C_p - q_i x (G Cpi)$$

Values of $q_z$ and $p$ for different heights are calculated using the above formulae and tabulated below.

<table>
<thead>
<tr>
<th>Height (ft)</th>
<th>$K_z$</th>
<th>$q_z$ (lb/ft$^2$)</th>
<th>$p$ (lb/ft$^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
<td>0.57471967</td>
<td>15.13214</td>
<td>13.246979</td>
</tr>
<tr>
<td>16</td>
<td>0.58541557</td>
<td>15.41376</td>
<td>13.4384798</td>
</tr>
<tr>
<td>17</td>
<td>0.59564407</td>
<td>15.68307</td>
<td>13.6216122</td>
</tr>
<tr>
<td>18</td>
<td>0.60545138</td>
<td>15.94129</td>
<td>13.7972035</td>
</tr>
<tr>
<td>19</td>
<td>0.61487687</td>
<td>16.18946</td>
<td>13.9659587</td>
</tr>
<tr>
<td>20</td>
<td>0.62395439</td>
<td>16.42847</td>
<td>14.1284837</td>
</tr>
</tbody>
</table>

Procedure Used by STAAD.Pro for Calculating the Equivalent Joint Load from incident Wind Pressure

1. Form closed panels. A closed panel is a region whose boundary consists entirely of members or of members and the ground surface.
2. Find the center of gravity of each of the panels.
3. For each panel, draw straight lines from the center of gravity (CG) to the midpoint of the members that form the panel boundary. So, the panel region will now contain several quadrilaterals whose two sides are made of portions of the respective members (or the ground) and the other two sides are lines going from the CG to the midpoint of the corresponding members.
4. The area contained in any quadrilateral is allocated as the influence area for the node located at the meeting point of two members.
5. Multiply the influence area by the average wind pressure contained inside the influence area and by the exposure factor for the node. This will yield the concentrated wind force for the joint.

Calculation of Equivalent Joint Loads
Evidently, members AB, BG and GH along with the ground surface, forms the closed panel ABGH. P is the CG of the panel, located at 5 ft away from line AH and 5 ft away from line AB. So, influence area for joint A is AJPI. Area of AJPI = 5 x 5 = 25 ft².

Average intensity of wind pressure in this area AJPI = 13.246979 lb/ft²

Exposure factor = 1 (for all the nodes in the model)

So, equivalent joint load in joint A = Influence area x Avg. Intensity of wind pressure x Exposure

= 25 x 13.246979 x 1 = 331.174 lb = 0.331 kip

Joint B will be under influence of influence area BKQU and BKPJ. Area of each of those influence areas = 5 x 5 = 25 ft². Intensity of wind pressure in each of those influence areas = 13.246979 lb/ft². Hence, equivalent joint load for each of those areas = 0.331 kip

So, total joint load in joint B = 0.331 x 2 = 0.662 kip

Joint C will be under influence area CVQU. Average intensity of wind pressure in this influence area will be = \((13.246979 + 14.1284837) / 2 = 13.68773135\) lb/ft²

Exposure Factor = 1

Area of CVQU = 5 x 5 = 25 ft²

Hence, equivalent joint load at joint C = 342.193 lb ≈ 0.343 kip

Joint W will be under influence of area WXRZ and WVQZ. Area of both of these = 25 ft². Average intensity of wind pressure in both of the influence areas = Average intensity of wind in the influence area CVQU = 13.68773135 lb/ft². Hence, equivalent joint load on joint W = 2 x 0.343 kip = 0.686 kip

Joint G will be under the influence of area GZRM, GZQK, GMSL and GKPL. Area of each of these influence areas = 5 x 5 = 25 ft². Average intensity of wind pressure in each of these areas = average intensity of wind pressure in area BKQU = 13.246979 lb/ft². Hence, equivalent joint load on joint G = 4 x 0.331 = 1.324 kip

Joint H will be under the influence of areas HLPI and HTSL. Area of each of those influence areas = 5 x 5 = 25 ft². Intensity of wind pressure in each of those influence areas = 13.246979 lb/ft². Hence, equivalent joint load on joint H = 2 x 0.331 = 0.662 kip

Since the structure is symmetric about the line WH, the equivalent joint load at D = equivalent joint load at C = 0.343 kip, equivalent joint load at B = equivalent joint load at E = 0.662 kip and equivalent joint load at A = equivalent joint load at F = 0.331 kip.

**Comparison**

**Table 475: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Joint load at node (kips)</td>
<td>1</td>
<td>0.331</td>
<td>0.331</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.662</td>
<td>0.662</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>0.662</td>
<td>0.662</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>0.324</td>
<td>0.325</td>
<td>negligible</td>
</tr>
<tr>
<td></td>
<td>9</td>
<td>0.331</td>
<td>0.331</td>
<td>none</td>
</tr>
<tr>
<td>Parameter</td>
<td>Hand Calculation</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>10</td>
<td>0.662</td>
<td>0.662</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>0.343</td>
<td>0.343</td>
<td>None</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>0.686</td>
<td>0.687</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>0.343</td>
<td>0.343</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\ASCE 7-10 Wind Load Generation.STD is typically installed with the program.

**STAAD SPACE**

START JOB INFORMATION

ENgINEER DATE 05-Sep-18

END JOB INFORMATION

*****************************************************************
*This problem has been created to verify the program calculated
*intensities of wind pressure as per ASCE 07-2010 and
*verify equivalent joint loads in the joints of a
*closed structure subjected to wind
*****************************************************************

INPUT WIDTH 79

UNIT FEET KIP

JOINT COORDINATES

1 0 0 0; 2 10 0 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10; 9 0 0 20; 10 0 10 20; 11 10 10 20; 12 10 0 20; 13 0 20 0;
14 10 20 0; 15 0 20 10; 16 10 20 10; 17 0 20 20; 18 10 20 20;

MEMBER INCIDENCES

1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11;
11 9 10; 12 10 11; 13 11 12; 14 2 13; 15 3 14; 16 6 15; 17 7 16; 18 10 17;
19 11 18; 20 13 14; 21 13 15; 22 14 16; 23 15 16; 24 15 17; 25 16 18; 26 17
18;

DEFINE MATERIAL START

ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL

MEMBER PROPERTY AMERICAN
1 TO 26 TABLE ST W14X873

CONSTANTS

MATERIAL STEEL_50_KSI ALL

SUPPORTS

1 4 5 8 9 12 FIXED

DEFINE WIND LOAD

TYPE 1 WIND 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
V. Wind On Open Structure

Validate the calculated member loads for an open structure subject to wind loading.

Details

A structure is subjected to wind loading from +X direction. The members are rectangular members with 500 mm × 500 mm in dimension. The diagonal member is also of same dimension but rotated at a beta angle value of 45°. Thus, the wind facing area of that member will be different than that of the other members.
Table 476: Variation of wind intensity over structure height

<table>
<thead>
<tr>
<th>Height (m)</th>
<th>Intensity of wind load (kN/m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>≤5</td>
<td>1.5</td>
</tr>
<tr>
<td>5 - 10</td>
<td>2</td>
</tr>
<tr>
<td>10 - 20</td>
<td>3</td>
</tr>
</tbody>
</table>

Validation

When open structures are subjected to wind load, wind load incident on the structure is applied as uniformly distributed loads on the members along the projected axis (i.e., in PX/PY/PZ direction). Wind intensity multiplied by the effective area of the member facing the wind load gives the uniformly distributed member load on the member.

Wind Load on Members 1, 3, 6 & 8

Wind intensity varies between the length of the member, so different parts of the member will be subjected to uniformly distributed loads of different magnitudes.

Let the midpoint of member 1 be K. So, $AK = CK = 5\ m$
Up to Height 5 m (part AK)

Intensity of wind load = 1.5 kN/m^2
Area of the member facing the wind = 0.5 m
\( w_{AK} = 1.5 \times 0.5 = 0.75 \text{ kN/m} \)

Up to Height 10 m (part KB)

Intensity of wind load = 2 kN/m^2
Area of the member facing the wind = 0.5 m
\( w_{KB} = 2 \times 0.5 = 1 \text{ kN/m} \)

Due to same orientation, member load on member 3, 6, and 8 will also be same as that of member 1.

Wind Load on Member 9, 10, 11 & 12

Intensity of wind load = 3 kN/m^2
Area of the member facing the wind = 0.5 m
\( w = 3 \times 0.5 = 1.5 \text{ kN/m} \)

Due to same orientation, member load on member 10, 11, and 12 will also be same as that of member 9.

Wind Load on Member 4 & 5

Intensity of wind load = 2 kN/m^2
Area of the member facing the wind = 0.5 m
\( w = 2 \times 0.5 = 1.0 \text{ kN/m} \)

Due to same orientation, member load on member 5 will also be same as that of member 4.

Wind Load on Member 14 & 15

Intensity of wind load = 3 kN/m^2
Area of the member facing the wind = 0.5 m
\( w = 3 \times 0.5 = 1.5 \text{ kN/m} \)

Due to same orientation, member load on member 15 will also be same as that of member 14.

Wind Load on Member 2, 7, 13, & 16

These members are aligned along the +X axis. Since the structure is subjected to wind load from +X direction only, the area of those members facing the wind is zero. So, there will be no load due to wind on these members.

Wind Load on Member 17

Since the member is rotated at a beta angle of 45°, the area of the member facing the wind = \( 0.5 \text{ m} / \cos 45° = 0.707 \text{ m} \)

Wind intensity varies between the length of the member, so different parts of the member will be subjected to uniformly distributed loads of different magnitudes.

Up to height 5m

Wind intensity = 1.5 kN/m^2
\( w = 1.5 \times 0.707 = 1.061 \text{ kN/m} \)

Up to height 10m
Wind intensity \(= 2 \text{kN/m}^2\)
\[ w = 2 \times 0.707 = 1.414 \text{kN/m} \]

Results

Table 477: Comparison of uniformly distributed load due to wind from +X on each member (kN/m)

<table>
<thead>
<tr>
<th>Member No</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>1, 3, 6 &amp; 8</td>
<td>Up to 5m: 0.75 5m - 10m: 1</td>
<td>Up to 5m: 0.75 5m - 10m: 1</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>4 &amp; 5</td>
<td>1</td>
<td>1</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>9, 10, 11, 12, 13, &amp; 15</td>
<td>1.5</td>
<td>1.5</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>Up to 5m: 1.061 5m - 10m: 1.414</td>
<td>Up to 5m: 1.0607 5m - 10m: 1.4142</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

Note: Members with no loading under this load condition are not displayed.

STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\Wind On Open Structure.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 30-Aug-18
END JOB INFORMATION
******************************************************************************
* This problem is created to determine the uniformly distributed load
* on the members when the structure is subjected to wind load
******************************************************************************
*
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10; 9 0 20 0; 10 10 20 0; 11 0 20 10; 12 10 20 10;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 2 5;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 17 PRIS YD 0.5 ZD 0.5
CONSTANTS
BETA 45 MEMB 17
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
DEFINE WIND LOAD
TYPE 1 WIND 1
INT 1.5 2 3 HEIG 5 10 20
EXP 1 JOINT 1 TO 12
LOAD 1 LOADTYPE Wind TITLE LOAD CASE 1
WIND LOAD X 1 TYPE 1 OPEN
PERFORM ANALYSIS PRINT LOAD DATA
FINISH

STAAD.Pro Output

LOADING 1 LOADTYPE WIND TITLE LOAD CASE 1
---------
MEMBER LOAD - UNIT KN METE
MEMBER UDL L1 L2 CON L LIN1 LIN2
1 0.7500 PX 0.00 5.00
1 1.0000 PX 5.00 10.00
3 0.7500 PX 5.00 10.00
3 1.0000 PX 0.00 5.00
4 1.0000 PX 0.00 10.00
5 1.0000 PX 0.00 10.00
6 0.7500 PX 0.00 5.00
6 1.0000 PX 5.00 10.00
8 0.7500 PX 5.00 10.00
8 1.0000 PX 0.00 5.00
9 1.0000 PX 0.00 0.00
9 1.5000 PX 0.00 10.00
10 1.0000 PX 0.00 0.00
10 1.5000 PX 0.00 10.00
11 1.0000 PX 0.00 0.00
11 1.5000 PX 0.00 10.00
12 1.0000 PX 0.00 0.00
12 1.5000 PX 0.00 10.00
14 1.5000 PX 0.00 10.00
15 1.5000 PX 0.00 10.00
17 1.0607 PX 7.07 14.14
17 1.4142 PX 0.00 7.07

V.07 Nonlinear Analysis
V. 2D Frame 2 Step P-Delta Displacement

To verify P-Delta analysis for a ten story plane frame.

Reference

Problem
Find the lateral displacement of the 10th story after two iterations of P-Delta analysis.

\[ E = 29,000 \text{ ksi} \]

\[ 123' 9@ 12' = 108' \]
\[ 15' 3@20' = 60' \]

*Figure 421: Ten story, plane frame*

Comparison

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement in X dir. at top (Node 41), ( \delta ) (in)</td>
<td>8.508</td>
<td>8.65347</td>
<td>1.7%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\07 Nonlinear Analysis\2D Frame 2 Step P-Delta Displacement.STD is typically installed with the program.

STAAD PLANE : 2-ITERATION PDELTA ANALYSIS OF A 10 STOREY PLANE FRAME
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
INPUT WIDTH 72
UNIT FEET KIP
JOINT COORDINATES
MEMBER INCIDENCES
1 1 5; 2 2 6; 3 3 7; 4 4 8; 5 5 6; 6 6 7; 7 7 8; 8 5 9; 9 6 10; 10 7 11;
11 8 12; 12 9 10; 13 10 11; 14 11 12; 15 9 13; 16 10 14; 17 11 15;
18 12 16; 19 13 14; 20 14 15; 21 15 16; 22 13 17; 23 14 18; 24 15 19;
25 16 20; 26 17 18; 27 18 19; 28 19 20; 29 17 21; 30 18 22; 31 19 23;
32 20 24; 33 21 22; 34 22 23; 35 23 24; 36 21 25; 37 22 26; 38 23 27;
39 24 28; 40 25 26; 41 26 27; 42 27 28; 43 25 29; 44 26 30; 45 27 31;
46 28 32; 47 29 30; 48 30 31; 49 31 32; 50 29 33; 51 30 34; 52 31 35;
53 32 36; 54 33 34; 55 34 35; 56 35 36; 57 33 37; 58 34 38; 59 35 39;
60 36 40; 61 37 38; 62 38 39; 63 39 40; 64 37 41; 65 38 42; 66 39 43;
67 40 44; 68 41 42; 69 42 43; 70 43 44;

MEMBER PROPERTY AMERICAN
61 TO 63 68 TO 70 TABLE ST W16X40
47 TO 49 54 TO 56 TABLE ST W21X44
33 TO 35 40 TO 42 TABLE ST W21X50
5 TO 7 12 TO 14 19 TO 21 26 TO 28 TABLE ST W24X55
1 4 8 11 TABLE ST W14X109
15 18 22 25 TABLE ST W14X82
29 32 36 39 TABLE ST W14X68
43 46 50 53 TABLE ST W14X53
57 60 64 67 TABLE ST W14X43
2 3 9 10 TABLE ST W14X159
16 17 23 24 TABLE ST W14X109
30 31 37 38 TABLE ST W14X90
44 45 51 52 TABLE ST W14X68
58 59 65 66 TABLE ST W14X48
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 TO 4 FIXED
UNIT FEET KIP
LOAD 1 LATERAL LOADS
SELFWEIGHT Y -1
JOINT LOAD
5 FX 2.97
9 FX 5.34
13 FX 7.71
17 FX 10.08
21 FX 12.45
V. Column Pushover Displacement

Compare theoretical solution to a pushover analysis to the STAAD.Pro solution.

Reference

Hand calculation.
Problem

A transverse load P is applied to a cantilever member and increased until the member fails.
Member length = 24 inch
Section = Wide flange W16X77
Material = Steel
Expected yield strength = 36 ksi

Figure 422: Cantilever member model

Figure 423: Plastic Rotation (in radians) vs. Moment (in kips) plot for moment hinge
Results from STAAD.Pro

**Note:** Pushover analysis requires the STAAD.Pro Advanced Analysis license.

The analysis results are saved at an interval of 0.1 inch deflection of the cantilever tip. The following results are displayed in the Postprocessing workflow by selecting the **Layouts > Pushover-Graphs** tool in the **Dynamics** group on the **Results** ribbon tab.

![Capacity curve (Displacement at Control Joint, inches vs. Base Shear, kips) as calculated by STAAD.Pro](image)

*Figure 424: Capacity curve (Displacement at Control Joint, inches vs. Base Shear, kips) as calculated by STAAD.Pro*

<table>
<thead>
<tr>
<th>Load Step</th>
<th>Displacement in</th>
<th>Base Shear kip</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0.153</td>
</tr>
<tr>
<td>2</td>
<td>0.001</td>
<td>1.153</td>
</tr>
<tr>
<td>3</td>
<td>0.078</td>
<td>57.837</td>
</tr>
<tr>
<td>4</td>
<td>0.136</td>
<td>61.953</td>
</tr>
<tr>
<td>5</td>
<td>0.243</td>
<td>63.979</td>
</tr>
<tr>
<td>6</td>
<td>0.349</td>
<td>65.979</td>
</tr>
<tr>
<td>7</td>
<td>0.455</td>
<td>67.979</td>
</tr>
<tr>
<td>8</td>
<td>0.562</td>
<td>69.979</td>
</tr>
<tr>
<td>9</td>
<td>0.668</td>
<td>71.979</td>
</tr>
</tbody>
</table>

*Table 479: Tabular form of the different points in Capacity curve as reported by STAAD.Pro*
<table>
<thead>
<tr>
<th>Load Step</th>
<th>Displacement in</th>
<th>Base Shear kip</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.774</td>
<td>73.979</td>
</tr>
<tr>
<td>11</td>
<td>0.880</td>
<td>75.979</td>
</tr>
<tr>
<td>12</td>
<td>0.986</td>
<td>77.979</td>
</tr>
<tr>
<td>13</td>
<td>1.065</td>
<td>79.479</td>
</tr>
<tr>
<td>14</td>
<td>1.065</td>
<td>21.510</td>
</tr>
</tbody>
</table>

Results from hand calculation

**At Load Step 3**

Equation of elastic deflection at cantilever tip:

\[ \delta_z = \frac{PL}{3EI} + \frac{PL}{GA_v} \]

where

\[
\begin{align*}
P &= 57.837 \text{ kip} \\
L &= 24 \text{ in.} \\
E &= 29,000 \text{ ksi} \\
I &= 138.0 \text{ in.}^4 \\
G &= 11,154 \text{ ksi} \\
A_v &= 10.4323 \text{ in.}^2
\end{align*}
\]

Thus elastic deformation:

\[
\delta_z = \frac{57.837 \times 24}{3 \times 29,000 \times 138} + \frac{57.837 \times 24}{11,154 \times 10.4323} = 0.066 + 0.012 = 0.078 \text{ in.}
\]

**At Load Step 13**

Equation of elastic deflection at cantilever tip

\[ \delta_z = \frac{PL}{3EI} + \frac{PL}{GA_v} \]

where

\[
\begin{align*}
P &= 79.479 \text{ kip}
\end{align*}
\]

(The other values are as listed for Load Step 3)

Thus elastic deformation at cantilever tip:

\[
\delta_z = \frac{79.479 \times 24}{3 \times 29,000 \times 138} + \frac{79.479 \times 24}{11,154 \times 10.4323} = 0.092 + 0.016 = 0.108 \text{ in.}
\]

Plastic rotation = 0.04 radian.

Since STAAD.Pro assumes small displacements (\(\sin \theta = \theta\)), plastic deformation at cantilever tip

\[ \delta_z \text{ plastic} = L \times \theta = 24 \times 0.04 = 0.96 \text{ inch} \]

Total deflection = \( \delta_z \text{ elastic} + \delta_z \text{ plastic} = 0.108 + 0.96 = 1.068 \text{ inch} \)
Comparison

Table 480: Comparison of results

<table>
<thead>
<tr>
<th>Load Step</th>
<th>Force P (free end) kips</th>
<th>Deflection (free end) in %</th>
<th>Percent Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>STAAD.Pro Advanced Analysis</td>
<td>Hand Calculation</td>
</tr>
<tr>
<td>3</td>
<td>57.837</td>
<td>0.078</td>
<td>0.078</td>
</tr>
<tr>
<td>13</td>
<td>79.479</td>
<td>1.065</td>
<td>1.068</td>
</tr>
</tbody>
</table>

Note: The deflection results from STAAD.Pro shown in the above table were obtained in the Postprocessing workflow.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\07 Nonlinear Analysis\Column Pushover Displacement.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Jul-07
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 24 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN 1 TABLE ST W16X77
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
DEFINE PUSHOVER DATA
FRAME 2
FYE 36.000000 ALL
HINGE PROPERTY MOMENT
TYPE 1
A 0 0 B 1 1 C 11 1.3 D 11 0.32 E 15 0.32 YM 1480 YR 0.004
IO 2 LS 7 CP 10.5
Hinge_type_1
Gnonl 1
Disp Z 2 Joint 2
Loading pattern 1
Ldstep 1
Spectrum parameters
Damping 2.0000
Sc 4
Ss 1.1
Si 1.6
Save loadstep result disp 0.10000
End pushover data
Load 1 Loadtype Gravity
Selfweight Z 1
Load 2 Loadtype Push
Joint load
2 Fz 1
Perform pushover analysis
Finish

STAAD Output

Problem Statistics
-----------------------------------
Number of joints          2  Number of members       1
Number of plates          0  Number of solids        0
Number of surfaces        0  Number of supports      1

Using 64-bit analysis engine.
Solver used is the in-core advanced math solver
Total primary load cases = 2, total degrees of freedom = 6
Total load combination cases = 0 so far.
More modes were requested than there are free masses.
Number of modes requested = 6
Number of existing masses in the model = 3
Number of modes that will be used = 3

*** EigenSolution : Advanced Method ***

Staad Space

Calculated frequencies for load case 3

Mode  Frequency(Cycles/Sec)  Period(SEC)
1.0  306.367  0.00326
2.0  544.518  0.00184
3.0  1865.405  0.00054

Modal weight (modal mass times g) in kip

Mode  X   Y   Z   Weight
1.0  0.000000E+00  0.000000E+00  7.674960E-02  7.674960E-02
2.0  7.674960E-02  0.000000E+00  0.000000E+00  7.674960E-02
3.0  0.000000E+00  7.674960E-02  0.000000E+00  7.674960E-02

Mass participation factors

Mode  X   Y   Z   Summ-X  Summ-Y  Summ-Z
1.0  0.00  0.00  100.00  0.000  0.000  100.000
2.0  100.00  0.00  0.00  100.000  0.000  100.000
3.0  0.00  100.00  0.00  100.000  100.000  100.000

*** Warning: Member # 1 has failed in " deformation-controlled

STAAD.Pro 3591 User Manual
V.08 Dynamic Analysis

V. Natural Frequency of a 2D Truss

To calculate the Natural frequency of vibration for a two story truss.

Reference
Hand calculation using known formula.

Problem
Find the natural frequency of vibration, $f$, of the truss.

Figure 425: Space truss

E = 30,000 ksi
L = 10 ft
P = 10 kips
All members are L4x4x5/16
Hand Calculation

Calculate average deflection at top:

$$\delta_{avg} = \frac{(0.45255 \text{ in} + 0.43535 \text{ in})}{2} = 0.444 \text{ in}$$

Stiffness of truss:

$$k = \frac{P}{\delta_{avg}} = \frac{20.0 \text{ kip}}{0.444 \text{ in}} = 45.5 \text{ k/in}$$

Mass:

$$M = \frac{w}{g} = \frac{20.0}{386.4} = 0.05175 \text{ k-sec}^2/\text{in}$$

Frequency:

$$f = \frac{\sqrt{k/m}}{2\pi} = \frac{\sqrt{45.057/0.05175}}{2\pi} = 4.696 \text{ Hz}$$

Comparison

Table 481: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency, $$f$$ (Hz)</td>
<td>4.696</td>
<td>4.687</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

STAAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Natural Frequency of a 2D Truss.STD is typically installed with the program.

STAAD TRUSS : A TWO STORY TRUSS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 0 20 0; 4 10 0 0; 5 10 10 0; 6 10 20 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 4 5; 4 5 6; 5 1 4; 6 2 5; 7 3 6; 8 2 4; 9 2 6;
MEMBER PROPERTY AMERICAN
1 TO 9 TABLE ST L40405
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
  E 4.176e+06
  POISSON 0.3
  DENSITY 0.489024
  ALPHA 6.5e-06
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL_50_KSI ALL
V. First Modal Frequency of a Cantilever Beam

To calculate the Natural frequency of vibration for a rectangular cantilever beam with a mass at the free end.

Reference
Hand calculation using known formulas.

Problem
Find the natural frequency of vibration, \( \omega \), of the cantilever beam.
Figure 426: Model for dynamic beam no. 1

- E = 30,000 ksi
- h = 12.0 in
- b = 6.0 in
- W = 10.0 k
- L = 120 in

**Hand Calculations**

Stiffness at free end:
\[
k = \frac{3EI}{L^3} = 45 \text{ k/in}
\]

Mass
\[
m = \frac{w}{g} = \frac{10.0 \text{ k}}{386.4 \text{ k·sec}^2/\text{in}} = 0.02588 \text{ k·sec}^2/\text{in}
\]

Circular frequency:
\[
\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{45}{0.02588}} = 41.7 \text{ rad/sec}
\]
\[
= 6.637 \text{ cycles/sec}
\]

**Comparison**

**Table 482: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency, f (Hz)</td>
<td>6.637</td>
<td>6.633</td>
<td>none</td>
</tr>
</tbody>
</table>
**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\First Modal Frequency of a Cantilever Beam.STD is typically installed with the program.

STAAD PLANE A RECTANGULAR CANTILEVER BEAM WITH A MASS AT THE FREE END

START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

INPUT WIDTH 72

SET SHEAR

UNIT FEET KIP

JOIN COORDINATES
1 0 0 0; 2 5 0 0; 3 10 0 0;

MEMBER INCIDENCES
1 1 2; 2 2 3;

UNIT INCHES KIP

MEMBER PROPERTY AMERICAN
1 2 PRIS YD 12 ZD 6

UNIT FEET KIP

DEFINE MATERIAL START

ISOTROPIC MATERIAL1
E 4.32e+06
POISSON 0.290909

END DEFINE MATERIAL

UNIT INCHES KIP

CONSTANTS

MATERIAL MATERIAL1 ALL

SUPPORTS
1 FIXED

CUT OFF MODE SHAPE 1

UNIT FEET KIP

LOAD 1 VERTICAL LOAD

MEMBER LOAD
2 CON Y -10 5
1 2 UNI GY -0.001

MODAL CALCULATION REQUESTED

PERFORM ANALYSIS

*PRINT MODE SHAPES

FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6.633</td>
<td>0.15076</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000000E+00</td>
<td>1.000514E+01</td>
<td>0.000000E+00</td>
<td>1.000299E+01</td>
</tr>
</tbody>
</table>

MASS PARTICIPATION FACTORS

-----------------------------
V. Rayleigh Natural Frequency of a Cantilever Beam

To calculate the Natural frequency of vibration using the Rayleigh method for a light cantilever beam with a mass at the free end.

Reference


Problem

Find the natural frequency of vibration, \( f \), of a mass, \( m \), attached to the end of a light cantilever beam of length, \( L \), and flexural stiffness, \( EI \).

\[
E = 30,000 \text{ ksi} \\
I = 1.3333 \text{ in}^4 \\
m = 0.1 \text{ lb-sec}^2/\text{in} \\
L = 30 \text{ in}
\]

Comparison

Table 483: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency, ( f ) (Hz)</td>
<td>33.553</td>
<td>33.5365</td>
<td>none</td>
</tr>
</tbody>
</table>
### V. Natural Frequency of a Simply Supported Beam

To find the fundamental frequency of vibration for a simply supported beam with a uniform mass.

**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Rayleigh Natural Frequency of a Cantilever Beam.STD` is typically installed with the program.

```
**STAAD Plane Natural Frequency of a Cantilevered Mass**

**START JOB INFORMATION**
ENGINEER DATE 14-Sep-18
**END JOB INFORMATION**
*  
* REFERENCE: THOMSON, W.T., "VIBRATION THEORY AND APPLICATIONS",  
* PRENTICE HALL INC., ENGLEWOODS, NEW JERSEY, 1965
*

**INPUT WIDTH 72**
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 PRIS IZ 1.33333
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1 NATURAL FREQUENCY
JOINT LOAD
2 FY -38.64
CALCULATE RAYLEIGH FREQUENCY
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
FINISH
```

**STAAD Output**

```
**********************************************************  
*                                                            
* RAYLEIGH FREQUENCY FOR LOADING 1 = 33.53649 CPS   *  
* MAX DEFLECTION = 0.00870 INCH GLO Y, AT JOINT 2   *  
*                                                            
**********************************************************  
```

---

**STAAD.Pro** 3598 **User Manual**
Reference

Problem
Find the fundamental frequency, $f$, of a simply supported beam of uniform cross-section.

\[ E = 30,000 \text{ ksi} \]
\[ I = 1.3333 \text{ in}^4 \]
\[ A = 4 \text{ in}^2 \]
\[ w = 1.124 \text{ lb/in} \]
\[ L = 80 \text{ in} \]

Figure 428: Model for dynamic beam no. 3

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>ANSYS®</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency, $f$ (Hz)</td>
<td>28.766</td>
<td>28.767</td>
<td>28.7438 (Rayleigh)</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>28.761 (Eigensolution)</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Natural Frequency of a Simply Supported Beam.STD is typically installed with the program.

START PLANE :FUNDAMENTAL FREQUENCY OF A SIMPLY SUPPORTED BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* REFERENCE: THOMSON, W.T., "VIBRATION THEORY AND APPLICATIONS", PRENTICE HALL INC., ENGLEWOODS, NEW JERSEY, 1965

INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 20 0 0; 3 40 0 0; 4 60 0 0; 5 80 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5;
MEMBER PROPERTY AMERICAN
1 TO 4 PRIS AX 4 IZ 1.33333
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 5 PINNED
CUT OFF FREQUENCY 100
CUT OFF MODE SHAPE 3
LOAD 1 DYNAMIC ANALYSIS (NATURAL FREQUENCY)
MEMBER LOAD
1 TO 4 UNI GY 1.124
CALCULATE RAYLEIGH FREQUENCY
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>CALCULATED FREQUENCIES FOR LOAD CASE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>28.761</td>
<td>0.03477</td>
</tr>
<tr>
<td>2</td>
<td>114.242</td>
<td>0.00875</td>
</tr>
<tr>
<td>3</td>
<td>242.560</td>
<td>0.00412</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MODAL WEIGHT (MODAL MASS TIMES g) IN POUN</th>
<th>GENERALIZED WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1                                     6.551152E+01 0.000000E+00 4.496000E+01</td>
<td></td>
</tr>
<tr>
<td>2                                     1.522398E-31 0.000000E+00 4.496000E+01</td>
<td></td>
</tr>
<tr>
<td>3                                     1.928479E+00 0.000000E+00 4.496000E+01</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MASS PARTICIPATION FACTORS IN PERCENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODE</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>3</td>
</tr>
</tbody>
</table>

* RAYLEIGH FREQUENCY FOR LOADING 1 = 28.74379 CPS *
* MAX DEFLECTION = 0.01499 INCH GLO Y, AT JOINT 3 *
V. Modal Frequencies of a Cantilever Beam

To find the first three natural frequencies of vibration for a cantilever beam with a uniform mass.

Reference


Problem

Find the first three natural frequencies, $f_1$, $f_2$, and $f_3$ of the cantilever beam.

![Model for dynamic beam no. 4](image)

**Figure 429: Model for dynamic beam no. 4**

- $E = 30,000$ ksi
- $I = 1.3333$ in$^4$
- $A = 4$ in$^2$
- $w = 1.124$ lb/in
- $L = 80$ in

Comparison

<table>
<thead>
<tr>
<th>Table 485: Comparison of results</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Result Type</strong></td>
</tr>
<tr>
<td>Frequency, $f_1$ (Hz)</td>
</tr>
</tbody>
</table>
**Result Type** | **Theory** | **ANSYS®** | **STAAD.Pro** | **Difference**
---|---|---|---|---
Frequency, $f_2$ (Hz) | 64.221 | 64.197 | 63.974 | none
Frequency, $f_3$ (Hz) | 179.82 | 180.14 | 178.672 | <1%

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Frequencies of a Cantilever Beam.STD is typically installed with the program.

```
STAAD PLAN : FREQUENCIES OF A CANTILEVERED BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE: THOMSON, W.T., "VIBRATION THEORY AND APPLICATIONS",
* PRENTICE HALL INC., ENGLEWOODS, NEW JERSEY, 1965
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 8 0 0; 4 12 0 0; 5 16 0 0; 6 20 0 0; 7 24 0 0;
8 28 0 0; 9 32 0 0; 10 36 0 0; 11 40 0 0; 12 44 0 0; 13 48 0 0;
14 52 0 0; 15 56 0 0; 16 60 0 0; 17 64 0 0; 18 68 0 0; 19 72 0 0;
20 76 0 0; 21 80 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 4 IZ 1.3333
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
CUT OFF FREQUENCY 200
CUT OFF MODE SHAPE 3
LOAD 1 UNIFORM MASS FOR MODAL ANALYSIS
MEMBER LOAD
1 TO 20 UNI GY 1.124
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```
V. Natural Frequency of Beam on Springs

Find the period of free vibration for a beam supported on two springs with a point mass.

Reference


Problem

A simple beam is supported by two spring as shown in the figure. Neglecting the distributed mass of the beam, calculate the period of free vibration of the beam subjected to a load of $W$.

\[
\begin{align*}
EI &= 30,000.0 \text{ ksi} \\
A &= 7.0 \text{ ft} \\
B &= 3.0 \text{ ft.} \\
W &= 1,000 \text{ lbf} \cdot K = 300.0 \text{ lb/in.}
\end{align*}
\]

![Figure 430: Beam supported on springs](image-url)
Comparison

Table 486: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Period (sec)</td>
<td>0.533</td>
<td>0.53317</td>
<td>negligible</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro\CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Natural Frequency of Beam on Springs.STD is typically installed with the program.

STAAD Plane: NATURAL FREQUENCY OF BEAM ON SPRINGS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE 'VIBRATION PROBLEMS IN ENGINEERING' BY
*  TIMOSHENKO, YOUNG, WEAVER. (4TH EDITION, PAGE 11, PROB 1.1-3)
*  THE ANSWER IN THE BOOK IS T = 0.533 sec., viz., F = 1.876 CPS
*
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 7 0 0; 3 10 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
UNIT INCHES POUND
SUPPORTS
1 3 FIXED BUT MZ KFY 300
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 1 IZ 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 1
LOAD 1 1000 LB LOAD AT JOINT 2
JOINT LOAD
2 FY -1000
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
V. Modal Frequencies of a Simply Supported Beam

To find the natural frequencies of vibration for a simply supported beam.

Reference


Problem

Find the first five flexural natural frequencies of the simple beam. Neglect shear deformation and rotary inertia.

\[
\begin{align*}
E &= 10,000 \text{ ksi} \\
\text{density} &= 0.1 \text{ lb/in}^3 \\
A_x &= 2.0 \text{ in}^2 \\
I_x &= 0.6667 \text{ in}^4 \\
L &= 20 \text{ in}
\end{align*}
\]

*Figure 431: Model for dynamic beam no. 7*
Hand Calculations

Weight

\[ w_{\text{weight}} = A \times \text{density} = 2.0 \times (0.1) = 0.2 \text{ lb/in} \]
\[ w_{\text{mass}} = 0.2 / (386.4) = 0.000518 \]

From Table 36, Item 1b of the reference:

\[ f_n = \frac{k_n}{2\pi} \sqrt{\frac{EI}{w_l^4}} = \frac{k_n}{2\pi} \sqrt{\frac{10(10)^3(0.6667)}{0.000518(20)^4}} = 45.16 \cdot k_n \]

Table 487: Modal stiffness and natural frequencies

<table>
<thead>
<tr>
<th>Mode</th>
<th>( k_n )</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.87</td>
<td>445.7</td>
</tr>
<tr>
<td>2</td>
<td>39.5</td>
<td>1,783.7</td>
</tr>
<tr>
<td>3</td>
<td>888</td>
<td>4,010.0</td>
</tr>
<tr>
<td>4</td>
<td>158</td>
<td>7,134.9</td>
</tr>
<tr>
<td>5</td>
<td>247</td>
<td>11,154</td>
</tr>
</tbody>
</table>

Comparison

Table 488: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency, ( f_1 ) (Hz)</td>
<td>445.7</td>
<td>445.495</td>
<td>none</td>
</tr>
<tr>
<td>Frequency, ( f_2 ) (Hz)</td>
<td>1,783.7</td>
<td>1,781.968</td>
<td>none</td>
</tr>
<tr>
<td>Frequency, ( f_3 ) (Hz)</td>
<td>4,010.0</td>
<td>4,009.310</td>
<td>none</td>
</tr>
<tr>
<td>Frequency, ( f_4 ) (Hz)</td>
<td>7,134.9</td>
<td>7,127.074</td>
<td>none</td>
</tr>
<tr>
<td>Frequency, ( f_5 ) (Hz)</td>
<td>11,154</td>
<td>11,133.978</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Frequencies of a Simply Supported Beam.STD is typically installed with the program.

STAAD PLANE : NATURAL FREQUENCIES OF A S.SUPPORTED BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE: W.C.YOUNG., "ROARK'S FORMULAS FOR STRESS &amp; STRAIN", 6TH ED.
* CASE 1B, TABLE 36, PAGE 714
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0;
9 8 0 0; 10 9 0 0; 11 10 0 0; 12 11 0 0; 13 12 0 0; 14 13 0 0;
15 14 0 0; 16 15 0 0; 17 16 0 0; 18 17 0 0; 19 18 0 0; 20 19 0 0;
21 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.33
DENSITY 0.1
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 2 IZ 0.666667
SUPPORTS
1 21 FIXED BUT MZ
2 TO 20 FIXED BUT FY MZ
CUT OFF MODE SHAPE 0
CUT OFF FREQUENCY 12000
LOAD 1
SELFWEIGHT Y -1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>445.495</td>
<td>0.00224</td>
</tr>
<tr>
<td>2</td>
<td>1781.968</td>
<td>0.00056</td>
</tr>
<tr>
<td>3</td>
<td>4009.310</td>
<td>0.00025</td>
</tr>
<tr>
<td>4</td>
<td>7127.074</td>
<td>0.00014</td>
</tr>
<tr>
<td>5</td>
<td>11133.978</td>
<td>0.00009</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MODE</th>
<th>MODAL WEIGHT (MODAL MASS TIMES g) IN POUND GENERALIZED</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X: 0.0000000E+00  Y: 3.228953E+00  Z: 0.0000000E+00</td>
</tr>
<tr>
<td>2</td>
<td>X: 0.0000000E+00  Y: 6.065180E-28  Z: 0.0000000E+00</td>
</tr>
<tr>
<td>3</td>
<td>X: 0.0000000E+00  Y: 3.469944E-01  Z: 0.0000000E+00</td>
</tr>
<tr>
<td>4</td>
<td>X: 0.0000000E+00  Y: 7.262741E-31  Z: 0.0000000E+00</td>
</tr>
<tr>
<td>5</td>
<td>X: 0.0000000E+00  Y: 1.165685E-01  Z: 0.0000000E+00</td>
</tr>
</tbody>
</table>

MASS PARTICIPATION FACTORS

--------------------------
| MASS PARTICIPATION FACTORS IN PERCENT |
--------------------------
V. Modal Response of a Beam

Find the natural frequencies for a beam and compare theoretical answers to the STAAD.Pro solution.

Reference

Problem
The first five natural frequencies and the associated mode shapes are computed for the flexural motion of a simply supported beam.

![Simple beam diagram](image1)

**Figure 432: Simple beam diagram**

L = 20 in

![Finite element model](image2)

**Figure 433: Finite element model**

The simply supported beam is divided into twenty spanwise beam elements. At nodes 1 and 21, all degrees of freedom except the rotation about the Z axis are restrained. For the remaining nodes, only the translation along Y and the rotation about Z are permitted. Both shear deformation and rotary inertia have been excluded from the model. The mass matrix is a diagonal matrix.

Cross-section Properties
Rectangular Section: 1 inch Width x 2 inch Depth

Area = 2 in²

\[
J = \frac{b^3}{16} \left\{ \frac{16}{3} + 3.36 \left( \frac{b}{a} \right) \left[ 1 - \frac{1}{12} \left( \frac{b}{a} \right)^4 \right] \right\}
\]

where

\[
\begin{align*}
a & = 2 \\
b & = 1
\end{align*}
\]
Theoretical Results

The natural bending frequencies, for a uniform beam with hinged ends, are given by:

\[ f_n = \frac{n^2 \pi^2}{2l^2} \sqrt{\frac{El}{A \gamma}} \]

where

- \( f_n \) = natural frequency for mode \( n \), in cycles per second
- \( l \) = span of the beam
- \( E \) = elastic modulus
- \( I \) = cross-section moment of inertia
- \( g \) = gravitational constant
- \( A \) = cross-section area
- \( \gamma \) = weight density

The parameters used in the frequency equation are:

- \( l = 20 \text{ in} \)
- \( E = 10 \times 10^6 \text{ psi} \)
- \( I = 0.6667 \text{ in}^4 \)
- \( g = 386.4 \text{ in}/s^2 \)
- \( A = 2.0 \text{ in}^2 \)
- \( \gamma = 0.1 \text{ lbs/in}^3 \)

from which:

\[ f_n = n^2 \times 445.686 \]

Comparison

The table below shows the natural frequencies computed from the theoretical equation and the subspace iteration method available within STAAD.Pro. Frequencies are in cycles per second.

**Table 489: Comparison of results**

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Theoretical</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>445.686</td>
<td>445.506</td>
<td>negligible</td>
</tr>
<tr>
<td>2</td>
<td>1,782.74</td>
<td>1,782.012</td>
<td>negligible</td>
</tr>
<tr>
<td>3</td>
<td>4,011.17</td>
<td>4,009.410</td>
<td>negligible</td>
</tr>
<tr>
<td>4</td>
<td>7,130.97</td>
<td>7,127.250</td>
<td>negligible</td>
</tr>
<tr>
<td>5</td>
<td>11,142.1</td>
<td>11,134.253</td>
<td>negligible</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a Beam.STD is typically installed with the program.

STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* Natural modes of a simple beam
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0;
9 8 0 0; 10 9 0 0; 11 10 0 0; 12 11 0 0; 13 12 0 0; 14 13 0 0;
15 14 0 0; 16 15 0 0; 17 16 0 0; 18 17 0 0; 19 18 0 0; 20 19 0 0;
21 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 2 IZ 0.6667
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
DENSITY 0.1
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 5
SUPPORTS
1 21 FIXED BUT MZ
2 TO 20 FIXED BUT FY MZ
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (CYCLES/SEC)</th>
<th>PERIOD (SEC)</th>
<th>MODAL WEIGHT (MODAL MASS TIMES g) IN POUNDS</th>
<th>GENERALIZED WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>445.506</td>
<td>0.00224</td>
<td>0.000000E+00 3.228953E+00 0.000000E+00</td>
<td>2.000000E+00</td>
</tr>
</tbody>
</table>
V. Modal Response of a Circular Plate

Find the natural frequencies of a circular plate and compare theoretical answers to the STAAD solution.

Reference

Problem
A flat circular plate is simply supported around the entire perimeter. The first six modes and their associated natural frequencies are to be computed using the subspace iteration method offered by STAAD. This problem demonstrates that the natural frequencies of an axi-symmetric structure can be accurately computed utilizing a 180 degree model with the appropriate boundary conditions.

The 180 degree sector was modeled using radial lines at intervals of 15 degrees. Tangential lines were then located utilizing a relationship such that the aspect ratio of the quad-plate elements was approximately 1.0. All rotations normal to the plane of the plate were restrained. In-plane translations for all nodes were restrained.
because the theoretical solution does not consider in-plane effects. Rotations about the global Y axis for the nodes at X=0.0 were restrained because this is a symmetry boundary. Z translation of all nodes on the outside radius were restrained to provide for the simply-supported condition.

\[ f_{ij} = \frac{\lambda_{ij}^2}{2na^2} \sqrt{\frac{Eh^3}{12\gamma(1-\nu^2)}} \]

= dimensionless parameter associated with the mode indices i, j

where

- \( i \) = number of nodal diameters in this mode shape
- \( j \) = number of nodal circles in this mode shape not counting the boundary
- \( \nu \) = Poisson's ratio
- \( E \) = elastic modulus
- \( h \) = plate thickness
- \( \gamma \) = mass of plate per unit area
- \( a \) = radius of plate

The numerical values used for this example are:

- \( \nu = 0.30 \)
- \( E = 10.0 \times 10^6 \) psi
\[ h = 0.10 \text{ inches} \]
\[ \gamma = \frac{(0.10 \text{ lb}_m \text{ in}^3)(0.10 \text{ in})}{(386.4 \text{ lb}_m \cdot \text{ in/lb}_f \cdot \text{s}^2)} = 2.588 \times 10^{-5} \text{ in/lb}_f \cdot \text{s}^2 \text{ in}^3 \]
\[ a = 10.0 \text{ inches} \]

with the numerical values used above

\[ \frac{1}{2n^2} \sqrt{\frac{E h^3}{12(1 - \nu^2)}} = \frac{1}{2n(10.0)^2} \sqrt{\frac{(10.0 \cdot 10^6)(0.10)^3}{12(2.588 \cdot 10^{-5})(1 - (0.3)^2)}} = 9.467 \text{ cycles/sec.} \]

\[ \lambda^2_{ij} \] is tabulated from the reference as follows:

**Table 490: Values of \( \lambda^2_{ij} \)**

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>( \lambda^2_{ij} )</th>
<th>Number of Nodal Diameters (i)</th>
<th>Number of Nodal Circles (j)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.977</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>13.94</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>25.65</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>29.76</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>5(1)</td>
<td>48.51</td>
<td>3</td>
<td>0</td>
</tr>
<tr>
<td>6</td>
<td>459.3</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

(1) not tabulated in the reference

**Comparison**

**Table 491: Comparison of results**

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Theoretical</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>47.12</td>
<td>46.211</td>
<td>1.9%</td>
</tr>
<tr>
<td>2</td>
<td>132.0</td>
<td>130.541</td>
<td>1.1%</td>
</tr>
<tr>
<td>3</td>
<td>242.8</td>
<td>240.146</td>
<td>1.1%</td>
</tr>
<tr>
<td>4</td>
<td>281.7</td>
<td>283.028</td>
<td>0.5%</td>
</tr>
<tr>
<td>5 *</td>
<td></td>
<td>373.197</td>
<td>n/a*</td>
</tr>
<tr>
<td>6</td>
<td>459.3</td>
<td>466.579</td>
<td>1.6%</td>
</tr>
</tbody>
</table>
*The reference did not tabulate a value of for the fifth mode of the structure, hence a comparison with the theoretical value of this mode cannot be made.

All anti-symmetric mode shapes for the 360 degree circular plate were captured by the 180 degree model with a phase angle included in the calculation. Some of the difference between the theoretical and STAAD.Pro frequencies is attributed to the loss of mass due to the piecewise secant representation of the outer radius and, since this mass is about 1 percent lower than for a true circular plate, it is not surprising that the first few modes are lower than the theoretical solution.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a Circular Plate.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* Natural Frequencies of a Circular Plate
UNIT INCHES POUND
JOINT COORDINATES
1 -6.39758e-07 -1 0; 2 0.258818 -0.965926 0; 3 0.5 -0.866026 0;
4 0.707107 -0.707107 0; 5 0.965926 -0.258819 0;
7 1 0 0; 8 0.965926 0.258818 0; 9 0.866025 -0.5 0;
10 0.707107 0.707107 0; 11 0.5 0.866026 0;
12 0.258818 0.965926 0;
13 -6.39758e-07 1 0; 14 -1.43562e-06 -2.244 0; 15 0.580789 -2.16754 0;
16 1.122 -1.94336 0; 17 1.58675 -1.58675 0; 18 1.94336 -1.122 0;
21 2.16754 -0.58079 0; 22 1.94336 1.122 0; 23 1.58675 1.58675 0;
24 1.122 1.94336 0;
25 0.58079 2.16754 0; 26 -1.43562e-06 -3.488 0; 27 -2.23148e-06 -3.488 0;
28 0.902759 -3.36915 0; 29 1.744 -3.0207 0;
30 2.46639 -2.46639 0; 31 3.0207 -1.744 0; 32 3.36915 -0.902761 0;
33 3.488 0 0; 34 3.36915 0.902761 0; 35 3.0207 1.744 0;
36 2.46639 2.46639 0; 37 1.744 3.0207 0; 38 0.902759 3.36915 0;
39 -2.23148e-06 3.488 0; 40 -2.90386e-06 -4.539 0; 41 1.17478 -4.38434 0;
42 2.2695 -3.93089 0; 43 3.20956 -3.20956 0;
44 3.93089 -2.2695 0; 45 4.38434 -1.17478 0; 46 4.539 0 0;
47 4.38434 1.17478 0; 48 3.93089 2.2695 0; 49 3.20956 3.20956 0;
50 2.2695 3.93089 0; 51 1.17478 4.38434 0; 52 -2.90386e-06 4.539 0;
53 -3.77841e-06 -5.906 0; 54 1.52858 -5.70476 0; 55 2.953 -5.11475 0;
56 4.17617 -4.17617 0; 57 5.11475 -2.953 0; 58 5.70476 -1.52859 0;
59 5.906 0 0; 60 5.70476 1.52859 0; 61 5.11475 2.953 0;
62 4.17617 4.17617 0; 63 2.953 5.11475 0; 64 1.52858 5.70476 0;
65 -3.77841e-06 5.906 0; 66 -4.91654e-06 -7.685 0;
67 1.98902 -7.42314 0; 68 3.8425 -6.65541 0; 69 5.43411 -5.43410 0;
70 6.6554 -3.8425 0; 71 7.42314 -1.98903 0; 72 7.685 0 0;
73 7.42314 1.98903 0; 74 6.6554 3.8425 0; 75 5.43411 5.43412 0;
76 3.8425 6.6554 1 0; 77 1.98902 7.42314 0; 78 -4.91654e-06 7.685 0;
79 -6.39758e-06 -10 0; 80 2.58818 -9.65926 0; 81 5 -8.66026 0;
82 7.07107 -7.07107 0; 83 8.66025 -5 0; 84 9.65926 -2.58819 0;
85 10 0 0; 86 9.65926 2.58819 0; 87 8.66025 5 0; 88 7.07107 7.07107 0;
89 5 8.66026 0; 90 2.58818 9.65926 0; 91 -6.39758e-06 10 0; 1000 0 0 0;
ELEMNET INCIDENCES SHELL
1 1000 1 2; 2 1000 2 3; 3 1000 3 4; 4 1000 4 5; 5 1000 5 6; 6 1000 6 7;
7 1000 7 8; 8 1000 8 9; 9 1000 9 10; 10 1000 10 11; 11 1000 11 12;
12 1000 12 13; 13 1 14 15 2; 14 2 15 16 3; 15 3 16 17 4; 16 4 17 18 5;
ELEMENT PROPERTY
1 TO 84 THICKNESS 0.1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
DENSITY 0.1
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
* centre of circle
 1000 FIXED BUT FZ MX
* Interior nodes
 2 TO 12 15 TO 25 28 TO 38 41 TO 51 54 TO 64 67 TO 77 FIXED BUT FZ MX MY
* nodes along circumference
 79 TO 91 FIXED BUT MX MY
* nodes along diameter except the two at the ends of the diameter.
 1 13 14 26 27 39 40 52 53 65 66 78 FIXED BUT FZ MX
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (CYCLES/SEC)</th>
<th>PERIOD (SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>46.211</td>
<td>0.02164</td>
</tr>
<tr>
<td>2</td>
<td>130.541</td>
<td>0.00766</td>
</tr>
<tr>
<td>3</td>
<td>240.146</td>
<td>0.00416</td>
</tr>
<tr>
<td>4</td>
<td>283.028</td>
<td>0.00353</td>
</tr>
<tr>
<td>5</td>
<td>373.197</td>
<td>0.00268</td>
</tr>
<tr>
<td>6</td>
<td>466.579</td>
<td>0.00214</td>
</tr>
</tbody>
</table>

MODAL WEIGHT (MODAL MASS TIMES g) IN POUN

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>GENERALIZED WEIGHT</th>
</tr>
</thead>
</table>

VERIFICATION EXAMPLES
V.08 Dynamic Analysis
STAAD.Pro 3615 User Manual
V. Modal Response of a Rectangular Plate

Find the natural frequencies of a rectangular plate and compare theoretical answers to the STAAD.Pro solution.

Reference

Problem
A flat rectangular plate is simply supported on all four sides. The first six modes and their associated natural frequencies are to be computed for this structure using the subspace iteration method offered by STAAD.Pro. This problem also demonstrates that the mesh refinement can be chosen to accurately calculate modes of interest based on the expected mode shapes.

![Figure 436: Simply supported, rectangular plate](image)

L = 45 in., W = 30 in., t = 0.2 in.

A plate with an aspect ratio of 1.5 was used so that comparison could be made with theoretical results tabulated for plates in the reference. An equally spaced mesh was utilized in both the x and the y dimensions of the plate. The number of elements in each dimension was determined on the basis of the highest mode of interest. Since
the number of half-waves in the sixth mode is 3 in the length dimension and 2 in the width dimension, a node spacing of 3.75 inches results in each half-wave being represented by four elements which means that no element will be expected to deform in double curvature. The simply supported edge condition requires that translation normal to the plane of the plate be restrained for these edge nodes. Rotations normal to the plate were restrained for all nodes.

\begin{figure}
\centering
\includegraphics[width=\textwidth]{modelFinite.png}
\caption{Model: finite element meshed}
\end{figure}

Theoretical Calculations

From the reference case 16 in Table 11-4, the first six natural frequencies of the plate are described by the following equations:

\[ f_{ij} = \frac{\lambda_{ij}^2}{2na^2} \sqrt{\frac{Eh^3}{12\gamma(1 - \nu^2)}} \]

where

\begin{align*}
  i & = \text{number of half-waves in this mode shape along the horizontal axis} \\
  j & = \text{number of half-waves in this mode shape along the vertical axis} \\
  \nu & = \text{Poisson's ratio} \\
  E & = \text{elastic modulus} \\
  h & = \text{plate thickness} \\
  \gamma & = \text{mass of material per unit area} \\
  a & = \text{length of plate} \\
  b & = \text{width of plate}
\end{align*}

The numerical values used for this example are:

\begin{align*}
  \nu & = 0.30 \\
  E & = 30.0 \times 10^6 \text{ psi} \\
  h & = 0.2 \text{ inches} \\
  \gamma & = \left( \frac{0.282 \text{ lb}_m}{\text{in}^3} \right) \left( 0.20 \text{ in} \right) \left( \frac{386.4 \text{ lb}_m \cdot \text{in/lb_f} \cdot \text{s}^2}{\text{in}^3} \right) = 0.146 \times 10^{-4} \text{ in/lb_f} \cdot \text{s}^2 \\
  a & = 45.0 \text{ in} \\
  b & = 30.0 \text{ in}
\end{align*}
with the numerical values used above

\[
\frac{1}{2\pi a^2} \sqrt{\frac{Eh^3}{12(1-v^2)}} = \frac{1}{2\pi (10.0)^{2}} \sqrt{\frac{(30.0 \cdot 10^6)(0.20)^3}{12(1.460 \cdot 10^{-4})(1 - (0.3)^2)}} = 0.9644 \text{ cycles/sec.}
\]

\(\lambda^2_{ij}\) is tabulated from the reference as follows:

**Table 492: Values of \(\lambda^2_{ij}\)**

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>(\lambda^2_{ij})</th>
<th>Number of Half-Waves in Length (i)</th>
<th>Number of Half-Waves in Width (j)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>32.08</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>61.69</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>98.70</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>4</td>
<td>111.0</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>128.3</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>6</td>
<td>177.7</td>
<td>3</td>
<td>2</td>
</tr>
</tbody>
</table>

**Comparison**

**Table 493: Comparison of results**

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Theoretical</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>30.94</td>
<td>30.599</td>
<td>1.1%</td>
</tr>
<tr>
<td>2</td>
<td>59.49</td>
<td>58.724</td>
<td>1.3%</td>
</tr>
<tr>
<td>3</td>
<td>95.18</td>
<td>95.063</td>
<td>negligible</td>
</tr>
<tr>
<td>4</td>
<td>107.1</td>
<td>106.277</td>
<td>0.8%</td>
</tr>
<tr>
<td>5</td>
<td>123.7</td>
<td>122.092</td>
<td>1.3%</td>
</tr>
<tr>
<td>6</td>
<td>171.3</td>
<td>168.009</td>
<td>1.9%</td>
</tr>
</tbody>
</table>

As noted earlier, the node spacing was based on the highest mode of interest. It follows that the difference between the theoretical and STAAD.Pro frequencies generally increases with increasing mode sequence.
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a Rectangular Plate.STD is typically installed with the program.

* Natural frequencies of a rectangular plate

UNIT INCHES POUND

JOINT COORDINATES
1 0 0 0; 2 3.75 0 0; 3 7.5 0 0; 4 11.25 0 0; 5 15 0 0; 6 18.75 0 0; 7 22.5 0 0; 8 26.25 0 0; 9 30 0 0; 10 33.75 0 0; 11 37.5 0 0; 12 41.25 0 0; 13 45 0 0; 14 0 3.75 0; 15 3.75 3.75 0; 16 7.5 3.75 0; 17 11.25 3.75 0; 18 15 3.75 0; 19 18.75 3.75 0; 20 22.5 3.75 0; 21 26.25 3.75 0; 22 30 3.75 0; 23 33.75 3.75 0; 24 37.5 3.75 0; 25 41.25 3.75 0; 26 45 3.75 0; 27 0 7.5 0; 28 3.75 7.5 0; 29 7.5 7.5 0; 30 11.25 7.5 0; 31 15 7.5 0; 32 18.75 7.5 0; 33 22.5 7.5 0; 34 26.25 7.5 0; 35 30 7.5 0; 36 33.75 7.5 0; 37 37.5 7.5 0; 38 41.25 7.5 0; 39 45 7.5 0; 40 0 11.25 0; 41 3.75 11.25 0; 42 7.5 11.25 0; 43 11.25 11.25 0; 44 15 11.25 0; 45 18.75 11.25 0; 46 22.5 11.25 0; 47 26.25 11.25 0; 48 30 11.25 0; 49 33.75 11.25 0; 50 37.5 11.25 0; 51 41.25 11.25 0; 52 45 11.25 0; 53 0 15 0; 54 3.75 15 0; 55 7.5 15 0; 56 11.25 15 0; 57 15 15 0; 58 18.75 15 0; 59 22.5 15 0; 60 26.25 15 0; 61 30 15 0; 62 33.75 15 0; 63 37.5 15 0; 64 41.25 15 0; 65 45 15 0; 66 0 18.75 0; 67 3.75 18.75 0; 68 7.5 18.75 0; 69 11.25 18.75 0; 70 15 18.75 0; 71 18.75 18.75 0; 72 22.5 18.75 0; 73 26.25 18.75 0; 74 30 18.75 0; 75 33.75 18.75 0; 76 37.5 18.75 0; 77 41.25 18.75 0; 78 45 18.75 0; 79 0 22.5 0; 80 3.75 22.5 0; 81 7.5 22.5 0; 82 11.25 22.5 0; 83 15 22.5 0; 84 18.75 22.5 0; 85 22.5 22.5 0; 86 26.25 22.5 0; 87 30 22.5 0; 88 33.75 22.5 0; 89 37.5 22.5 0; 90 41.25 22.5 0; 91 45 22.5 0; 92 0 26.25 0; 93 3.75 26.25 0; 94 7.5 26.25 0; 95 11.25 26.25 0; 96 15 26.25 0; 97 18.75 26.25 0; 98 22.5 26.25 0; 99 26.25 26.25 0; 100 30 26.25 0; 101 33.75 26.25 0; 102 37.5 26.25 0; 103 41.25 26.25 0; 104 45 26.25 0; 105 0 30 0; 106 3.75 30 0; 107 7.5 30 0; 108 11.25 30 0; 109 15 30 0; 110 18.75 30 0; 111 22.5 30 0; 112 26.25 30 0; 113 30 30 0; 114 33.75 30 0; 115 37.5 30 0; 116 41.25 30 0; 117 45 30 0;

ELEMENT INCIDENCES SHELL
1 1 1 2 15 14; 2 2 3 16 15; 3 3 4 17 16; 4 4 5 18 17; 5 5 6 19 18; 6 6 7 20 19; 7 7 8 21 20; 8 8 9 22 21; 9 9 10 23 22; 10 10 11 24 23; 11 11 12 25 24; 12 12 13 26 25; 13 13 14 27 26; 14 14 15 28 27; 15 15 16 29 28; 16 16 17 30 29; 17 17 18 31 30; 18 18 19 32 31; 19 19 20 33 32; 20 20 21 34 33; 21 21 22 35 34; 22 22 23 36 35; 23 23 24 37 36; 24 24 25 38 37; 25 25 26 39 38; 26 26 27 40 39; 27 27 28 41 40; 28 28 29 42 41; 29 29 30 43 42; 30 30 31 44 43; 31 31 32 45 44; 32 32 33 46 45; 33 33 34 47 46; 34 34 35 48 47; 35 35 36 49 48; 36 36 37 50 49; 37 37 38 51 50; 38 38 39 52 51; 39 39 40 53 52; 40 40 41 54 53; 41 41 42 55 54; 42 42 43 56 55; 43 43 44 57 56; 44 44 45 58 57; 45 45 46 59 58; 46 46 47 60 59; 47 47 48 61 60; 48 48 49 62 61; 49 49 50 63 62; 50 50 51 64 63; 51 51 52 65 64; 52 52 53 66 65; 53 53 54 67 66; 54 54 55 68 67; 55 55 56 69 68; 56 56 57 70 69; 57 57 58 71 70; 58 58 59 72 71; 59 59 60 73 72; 60 60 61 74 73; 61 61 62 75 74; 62 62 63 76 75; 63 63 64 77 76; 64 64 65 78 77; 65 65 66 79 78; 66 66 67 80 79; 67 67 68 81 80; 68 68 69 82 81; 69 69 70 83 82; 70 70 71 84 83; 71 71 72 85 84;
ELEMENT PROPERTY
1 TO 96 THICKNESS 0.2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
DENSITY 0.282
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 6
CUT OFF FREQUENCY 1000
SUPPORTS
* Corner nodes
1 13 105 117 FIXED BUT MX MY
* Nodes along y=0 and y=30
2 TO 12 106 TO 116 FIXED BUT MX MY
* Nodes along x=0
14 27 40 53 66 79 92 FIXED BUT MX MY
* Nodes along x=45
26 39 52 65 78 91 104 FIXED BUT MX MY
* Interior nodes
15 TO 25 28 TO 38 41 TO 51 54 TO 64 67 TO 77 80 TO 90 93 TO 102 -
103 FIXED BUT FZ MX MY
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>30.599</td>
<td>0.03268</td>
</tr>
<tr>
<td>2</td>
<td>58.724</td>
<td>0.01703</td>
</tr>
<tr>
<td>3</td>
<td>95.063</td>
<td>0.01052</td>
</tr>
<tr>
<td>4</td>
<td>106.277</td>
<td>0.00941</td>
</tr>
<tr>
<td>5</td>
<td>122.092</td>
<td>0.00819</td>
</tr>
<tr>
<td>6</td>
<td>168.009</td>
<td>0.00595</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>GENERALIZED WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>4.833474E+01</td>
<td>1.915440E+01</td>
</tr>
<tr>
<td>2</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>9.476868E-17</td>
<td>1.920213E+01</td>
</tr>
<tr>
<td>3</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>4.308438E-14</td>
<td>1.925532E+01</td>
</tr>
<tr>
<td>4</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>4.837131E+00</td>
<td>1.927455E+01</td>
</tr>
</tbody>
</table>
V. Modal Response of a 3D Frame

Find the natural frequencies of a space frame. Compare the results from subspace iteration method used by STAAD.Pro with the results from EASE2® and ANSYS®.

Reference
1. Problem 1, from the ASME 1972 Program Verification and Qualification Library

Problem
A three dimensional frame is analyzed for its natural frequencies and the associated mode shapes using the subspace iteration method offered by STAAD.Pro.
Figure 438: Frame model

L1 = 8.625 ft
L2 = 10 ft

All dimensions are in inches.

The only entity present in the model is the general purpose three dimensional beam.

Cross-section Properties

All the beam elements have the same cross section:

Outside radius, \( R_o = 1.1875 \text{ in.} \)
Inside radius, \( R_i = 1.0335 \text{ in.} \)
Area, \( A = \pi (R_o^2 - R_i^2) = 1.074532 \text{ in}^2 \)
Moment of Inertia, \( I = \pi / 4 (R_o^4 - R_i^4) \)
\[ I = 0.665747 \text{ in}^4 \]
Torsion constant = twice the inertia, for closed circular sections = \( 1.331494 \text{ in}^4 \)
The expression for the shear flexibility factor is derived from the ratio of maximum shear stress to the average shear stress:

\[
\alpha = \frac{AQ}{Ib}
\]

Where the four items – A, Q, I, and b are properties of the half cross-section about the centerline. For the circular section they are:

\[
A = \frac{\pi}{2}(R_o^2 - R_i^2)
\]

\[
Q = \frac{2}{3}(R_o^3 - R_i^3)
\]

\[
I = \frac{\pi}{8}(R_o^4 - R_i^4)
\]

\[
b = 2(R_o - R_i)
\]

The final expression for the shear flexibility factor, for a circular tube section is:

\[
\alpha = \frac{A}{b} = \frac{4}{3} \frac{R_o^3 - R_i^3}{(R_o^2 - R_i^2)(R_o - R_i)}
\]

\[
\alpha = 1.993620
\]

Hence the shear area is entered as

\[
AY = \text{Cross Section Area} / \alpha = \frac{1.074532}{1.99362} = 0.538985 \text{ in}^2
\]

Comparison

The following table compares Twenty-Four Natural Frequencies

<table>
<thead>
<tr>
<th>Table 494: Comparison of results</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Mode Number</strong></td>
</tr>
<tr>
<td>-----------------</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>3</td>
</tr>
<tr>
<td>4</td>
</tr>
<tr>
<td>5</td>
</tr>
<tr>
<td>6</td>
</tr>
<tr>
<td>7</td>
</tr>
<tr>
<td>8</td>
</tr>
<tr>
<td>9</td>
</tr>
<tr>
<td>10</td>
</tr>
<tr>
<td>11</td>
</tr>
</tbody>
</table>
In both references by DeSalvo and Swanson\textsuperscript{2} and Peterson\textsuperscript{3}, the number of dynamic degrees of freedom has been reduced from 42 to 24, by means of the Guyan method. No such reduction is performed in the STAAD.Pro model.

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>DeSalvo and Swanson\textsuperscript{2}</th>
<th>Peterson\textsuperscript{3}</th>
<th>STAAD.Pro</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>894.08</td>
<td>894.26</td>
<td>892.281</td>
</tr>
<tr>
<td>13</td>
<td>910.21</td>
<td>910.40</td>
<td>893.060</td>
</tr>
<tr>
<td>14</td>
<td>916.98</td>
<td>917.18</td>
<td>910.829</td>
</tr>
<tr>
<td>15</td>
<td>940.02</td>
<td>940.02</td>
<td>932.286</td>
</tr>
<tr>
<td>16</td>
<td>959.98</td>
<td>960.27</td>
<td>956.391</td>
</tr>
<tr>
<td>17</td>
<td>971.15</td>
<td>971.44</td>
<td>964.012</td>
</tr>
<tr>
<td>18</td>
<td>976.92</td>
<td>977.22</td>
<td>967.267</td>
</tr>
<tr>
<td>19</td>
<td>1012.2</td>
<td>1012.5</td>
<td>981.536</td>
</tr>
<tr>
<td>20</td>
<td>1028.4</td>
<td>1028.8</td>
<td>1009.256</td>
</tr>
<tr>
<td>21</td>
<td>1123.6</td>
<td>1123.9</td>
<td>1070.562</td>
</tr>
<tr>
<td>22</td>
<td>1134.5</td>
<td>1134.9</td>
<td>1123.161</td>
</tr>
<tr>
<td>23</td>
<td>1164.1</td>
<td>1164.4</td>
<td>1149.368</td>
</tr>
<tr>
<td>24</td>
<td>1216.7</td>
<td>1217.2</td>
<td>1199.882</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a 3D Frame.STD is typically installed with the program.

STAAD SPACE

START JOB INFORMATION

ENGINEER DATE 14-Sep-18

END JOB INFORMATION

* Natural Modes of a space frame

UNIT INCHES POUND

JOINT COORDINATES

1 0 0 0; 2 27.25 0 0; 3 0 10 0; 4 27.25 10 0; 5 0 18.625 0;
6 8.625 18.625 0; 7 18.625 18.625 0; 8 27.25 18.625 0; 9 0 18.625 8.625;
10 27.25 18.625 8.625; 11 0 0 17.25; 12 27.25 0 17.25; 13 0 10 17.25;
14 27.25 10 17.25; 15 0 18.625 17.25; 16 8.625 18.625 17.25;
17 18.625 18.625 17.25; 18 27.25 18.625 17.25;

MEMBER INCIDENCES

1 1 3; 2 2 4; 3 3 5; 4 4 8; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 8 10; 10 9 15;
11 10 18; 12 11 13; 13 12 14; 14 13 15; 15 14 18; 16 15 16; 17 16 17;
18 17 18;
MEMBER PROPERTY AMERICAN
  1 TO 17 -
18 PRIS AX 1.07453 AY 0.538985 AZ 0.538985 IX 1.33149 IY 0.665747 -
IZ 0.665747
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.79e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 24
SUPPORTS
1 2 11 12 FIXED
LOAD 1
JOINT LOAD
  3 4 13 14 FX 3.4517 FY 3.4517 FZ 3.4517
  6 7 16 17 FX 3.4517 FY 3.4517 FZ 3.4517
  9 10 FX 3.4517 FY 3.4517 FZ 3.4517
  5 8 15 18 FX 9.7973 FY 9.7973 FZ 9.7973
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>111.236</td>
<td>0.00899</td>
</tr>
<tr>
<td>2</td>
<td>115.801</td>
<td>0.00864</td>
</tr>
<tr>
<td>3</td>
<td>137.169</td>
<td>0.00729</td>
</tr>
<tr>
<td>4</td>
<td>215.799</td>
<td>0.00463</td>
</tr>
<tr>
<td>5</td>
<td>404.270</td>
<td>0.00247</td>
</tr>
<tr>
<td>6</td>
<td>422.642</td>
<td>0.00237</td>
</tr>
<tr>
<td>7</td>
<td>451.599</td>
<td>0.00221</td>
</tr>
<tr>
<td>8</td>
<td>549.012</td>
<td>0.00182</td>
</tr>
<tr>
<td>9</td>
<td>733.589</td>
<td>0.00136</td>
</tr>
<tr>
<td>10</td>
<td>758.563</td>
<td>0.00132</td>
</tr>
<tr>
<td>11</td>
<td>851.354</td>
<td>0.00117</td>
</tr>
<tr>
<td>12</td>
<td>892.281</td>
<td>0.00112</td>
</tr>
<tr>
<td>13</td>
<td>893.060</td>
<td>0.00112</td>
</tr>
<tr>
<td>14</td>
<td>910.829</td>
<td>0.00110</td>
</tr>
<tr>
<td>15</td>
<td>932.286</td>
<td>0.00107</td>
</tr>
<tr>
<td>16</td>
<td>956.391</td>
<td>0.00105</td>
</tr>
<tr>
<td>17</td>
<td>964.012</td>
<td>0.00104</td>
</tr>
<tr>
<td>18</td>
<td>967.267</td>
<td>0.00103</td>
</tr>
<tr>
<td>19</td>
<td>981.536</td>
<td>0.00102</td>
</tr>
<tr>
<td>20</td>
<td>1009.256</td>
<td>0.00099</td>
</tr>
<tr>
<td>21</td>
<td>1070.562</td>
<td>0.00093</td>
</tr>
<tr>
<td>22</td>
<td>1123.161</td>
<td>0.00089</td>
</tr>
<tr>
<td>23</td>
<td>1149.368</td>
<td>0.00087</td>
</tr>
<tr>
<td>24</td>
<td>1199.882</td>
<td>0.00083</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>GENERALIZED WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6.984127E+01</td>
<td>6.404218E-24</td>
<td>1.284664E-24</td>
<td>6.127270E+01</td>
</tr>
</tbody>
</table>
### Verification Examples

**V.08 Dynamic Analysis**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Mass Participation Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1.547059E-22 1.789605E-20 7.005318E+01</td>
</tr>
<tr>
<td>3</td>
<td>1.064194E-21 1.226078E-19 9.255584E-03</td>
</tr>
<tr>
<td>4</td>
<td>9.147275E-20 1.045721E-17 1.002150E-20</td>
</tr>
<tr>
<td>5</td>
<td>4.553345E-19 4.244579E-03 4.985574E-20</td>
</tr>
<tr>
<td>6</td>
<td>5.447639E-22 1.375280E-18 5.112414E-03</td>
</tr>
<tr>
<td>7</td>
<td>1.160472E-20 1.825470E+01 1.949661E-25</td>
</tr>
<tr>
<td>8</td>
<td>1.055933E-23 9.521535E+00 1.626407E-22</td>
</tr>
<tr>
<td>9</td>
<td>9.808838E-04 8.794435E-20 3.656936E-22</td>
</tr>
<tr>
<td>10</td>
<td>1.321191E+00 9.714550E-18 1.273004E-20</td>
</tr>
<tr>
<td>11</td>
<td>6.082156E-21 1.911260E+00 1.769786E-20</td>
</tr>
<tr>
<td>12</td>
<td>1.722239E+00 1.063875E-16 3.802991E+01</td>
</tr>
<tr>
<td>13</td>
<td>4.611356E-20 4.549234E-17 5.528088E-21</td>
</tr>
<tr>
<td>14</td>
<td>1.400995E-20 6.381431E-17 2.363042E-01</td>
</tr>
<tr>
<td>15</td>
<td>8.893063E-19 1.393844E-18 1.393844E-18</td>
</tr>
<tr>
<td>16</td>
<td>1.758278E-18 2.780487E-15 3.083966E+00</td>
</tr>
<tr>
<td>17</td>
<td>3.491399E-01 1.038217E-15 5.994489E-19</td>
</tr>
<tr>
<td>18</td>
<td>5.049439E-19 1.122183E+00 4.036567E-19</td>
</tr>
<tr>
<td>19</td>
<td>3.350978E-18 3.238417E-15 1.516123E-18</td>
</tr>
<tr>
<td>20</td>
<td>8.666280E-17 9.060816E-14 3.752768E-17</td>
</tr>
<tr>
<td>21</td>
<td>6.382000E-02 1.876943E-13 5.827891E-17</td>
</tr>
<tr>
<td>22</td>
<td>9.476000E-02 1.976439E-15 5.994489E-19</td>
</tr>
<tr>
<td>23</td>
<td>5.049439E-19 1.122183E+00 4.036567E-19</td>
</tr>
<tr>
<td>24</td>
<td>3.350978E-18 3.238417E-15 1.516123E-18</td>
</tr>
</tbody>
</table>

**STAAD SPACE**

**MASS PARTICIPATION FACTORS**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Mass Participation Factors</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>94.76 0.00 0.00 94.756 0.000 0.000</td>
</tr>
<tr>
<td>2</td>
<td>0.00 95.04 0.00 94.756 0.000 95.044</td>
</tr>
<tr>
<td>3</td>
<td>0.00 0.00 0.00 94.756 0.000 95.044</td>
</tr>
<tr>
<td>4</td>
<td>0.00 0.00 0.00 94.756 0.000 95.044</td>
</tr>
<tr>
<td>5</td>
<td>0.00 0.00 0.00 94.756 0.006 95.044</td>
</tr>
<tr>
<td>6</td>
<td>0.00 24.77 0.00 94.756 24.773 95.044</td>
</tr>
<tr>
<td>7</td>
<td>0.00 0.00 0.01 94.756 24.773 95.044</td>
</tr>
<tr>
<td>8</td>
<td>0.00 0.00 0.40 94.756 24.773 95.043</td>
</tr>
<tr>
<td>9</td>
<td>0.00 12.92 0.00 94.756 37.691 95.043</td>
</tr>
<tr>
<td>10</td>
<td>0.00 0.00 0.00 94.756 37.691 95.043</td>
</tr>
<tr>
<td>11</td>
<td>1.79 0.00 0.00 96.550 37.691 95.043</td>
</tr>
<tr>
<td>12</td>
<td>0.00 2.59 0.00 96.550 40.284 95.043</td>
</tr>
<tr>
<td>13</td>
<td>2.34 0.00 0.00 98.887 40.284 95.043</td>
</tr>
<tr>
<td>14</td>
<td>0.00 0.00 0.00 98.887 40.284 95.043</td>
</tr>
<tr>
<td>15</td>
<td>0.00 0.00 0.32 98.887 40.284 95.043</td>
</tr>
<tr>
<td>16</td>
<td>0.00 5.59 0.00 98.887 45.878 95.043</td>
</tr>
<tr>
<td>17</td>
<td>0.00 0.00 0.00 98.887 45.878 95.043</td>
</tr>
<tr>
<td>18</td>
<td>0.00 0.00 4.18 98.887 45.878 99.958</td>
</tr>
<tr>
<td>19</td>
<td>0.47 0.00 0.00 99.360 45.878 99.958</td>
</tr>
<tr>
<td>20</td>
<td>0.00 15.23 0.00 99.360 61.103 99.958</td>
</tr>
<tr>
<td>21</td>
<td>0.00 0.00 0.00 99.360 61.103 99.958</td>
</tr>
<tr>
<td>22</td>
<td>0.49 0.00 0.00 99.854 61.103 99.958</td>
</tr>
<tr>
<td>23</td>
<td>0.00 0.00 0.00 99.854 61.103 99.958</td>
</tr>
<tr>
<td>24</td>
<td>0.09 0.00 0.00 99.940 61.103 99.958</td>
</tr>
</tbody>
</table>
V. Beam Subject to response spectrum

Find the maximum moment due to the time history loading and compare theoretical answers to the STAAD solution.

Reference


Problem

The supports of a simply supported beam are subjected to an acceleration time history. The maximum bending moment in the beam is computed for the first mode of the structure. This problem demonstrates the capabilities of STAAD to calculate the correct modal response of a structure utilizing response spectrum data.

![Figure 439: Simple span beam](image)

$L = 240$ in.

The STAAD model consists of 11 nodes and 10 elastic beam elements. Node 1 is completely restrained with the exception of having rotational freedom in the Z direction, the remaining nodes are restrained except for X and Y displacements and Z rotations. Node 11 is additionally restrained against displacements in the Y direction to provide for the simple support condition. Only the contribution of the first mode of the structure is considered.

![Figure 440: Finite element model](image)

**Theoretical Solution**

Material Properties

$E = 30 \times 10^6$ lb/in$^2$

$EI = 1.0 \times 10^{10}$ lb-in$^2$

$m = 0.2$ lb-sec$^2$/in$^2$

$h = 14.0$ in.

From Reference 2, Table 8-1, page 108, the fundamental frequency of the beam is:
\[ f_i = \frac{\lambda_i^2}{2n l^2} \sqrt{\frac{E I}{m}} = \frac{9.869}{2n(240)^2} \sqrt{\frac{1.0(10)^{10}}{0.2}} = 6.098 \text{Hz} \]

The modal participation factor for the fundamental mode is:

\[ \Gamma = \frac{\int_0^1 m \phi(x) \, dx}{\int_0^1 m \phi^2(x) \, dx} \]

Where the first mode shape, \( \phi(x) = \sin(\pi x / l) \)

\[ \Gamma = \frac{\int_0^1 m \sin \left( \frac{\pi x}{l} \right) \, dx}{\int_0^1 m \sin^2 \left( \frac{\pi x}{l} \right) \, dx} = \frac{4}{\pi} \]

The maximum relative modal displacement is given by:

\[ A_{\text{max}} = \Gamma u_{\text{max}} \]

where

\[ u_{\text{max}} = \frac{y''_{\text{so}}}{\omega^2 (\text{DLF})_{\text{max}}} \]
\[ \omega = 2\pi \times 6.098 \text{Hz} \]
\[ y''_{\text{so}} = 1.0g \]
\[ (\text{DLF})_{\text{max}} = 1.648 \text{ at } 6.098 \text{ Hz} \]

therefore:

\[ A_{\text{max}} = 4(1.648)(386.4) / [\pi(2\pi)^2(6.098)^2] = 0.5523 \text{ in.} \]

The bending moment

\[ M = -E I \frac{\delta^2 u}{(\delta x)^2} \]

Where \( u \) for the first mode = \( A \sin(\pi x / l) \)

\[ \delta^2 u / (\delta x)^2 = -\pi^2 l^2 A \sin(\pi x / l) \]
\[ M = A E I \left( \frac{\pi^2 l^2}{2} \right) \sin(\pi x / l) \]
\[ M_{\text{max}} = A_{\text{max}} E I \left( \frac{\pi^2 l^2}{2} \right) \]

at x = l/2

\[ M_{\text{max}} = 1(10)^{10} \times (0.5523) \times \pi^2 / (240^2) = 946.351(10)^3 \text{ lb in} \]

at x = l/2

**Comparison**

Table 495: Comparison of results

<table>
<thead>
<tr>
<th>Solution</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending Moment (kip-inch)</td>
<td>946.351</td>
<td>947.088</td>
<td>negligible</td>
</tr>
</tbody>
</table>
Verification Examples
V.08 Dynamic Analysis

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Beam Subject to response spectrum.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION

* RESPONSE OF A SIMPLY SUPPORTED BEAM TO A SHOCK SPECTRUM
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 24 0 0; 3 48 0 0; 4 72 0 0; 5 96 0 0; 6 120 0 0; 7 144 0 0; 8 168 0 0; 9 192 0 0; 10 216 0 0; 11 240 0 0;
MEMBER INCIDENCES
1 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10; 10 10 11;
MEMBER PROPERTY AMERICAN
1 TO 10 PRIS AX 20.4082 IX 40 IY 3.6139 IZ 333.333 YD 14 ZD 1.45777
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
DENSITY 3.78672
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 1
SUPPORTS
1 FIXED BUT MZ
2 TO 10 FIXED BUT FX FY MZ
11 FIXED BUT FX MZ
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SPECTRUM SRSS Y 1 ACC SCALE 386.4 DAMP 0.001
0.15 1.648; 0.17 1.648;
PERFORM ANALYSIS
PRINT MEMBER FORCES LIST 5
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>CALCULATED FREQUENCIES FOR LOAD CASE</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MODE</strong></td>
<td>1</td>
</tr>
<tr>
<td>FREQUENCY (CYCLES/SEC)</td>
<td>6.069</td>
</tr>
<tr>
<td>PERIOD (SEC)</td>
<td>0.16476</td>
</tr>
<tr>
<td>RESPONSE SPECTRUM LOAD</td>
<td>1</td>
</tr>
<tr>
<td>RESPONSE LOAD CASE</td>
<td>1</td>
</tr>
<tr>
<td>MODAL WEIGHT (MODAL MASS TIMES g) IN POUN</td>
<td>GENERALIZED</td>
</tr>
<tr>
<td>X Y Z WEIIGHT</td>
<td>9.273617E+03</td>
</tr>
<tr>
<td>SRSS MODAL COMBINATION METHOD USED.</td>
<td>1.761987E+04</td>
</tr>
<tr>
<td>DYNAMIC WEIGHT X Y Z</td>
<td>1.669251E+04</td>
</tr>
<tr>
<td>POUN</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>
V. Steady State Loading on a Beam

Calculate the deflections at two points along the beam at steady-state condition.

Reference

### Problem
Determine the steady-state displacements of the quarter and mid-span points of a fixed-fixed beam subjected to a parabolically varying distributed load operating at a 7.5 Hz frequency.

![Beam with harmonic distributed load](image)

**Figure 441: Beam with harmonic distributed load**

![Model: divide span 20@10.0”](image)

**Figure 442: Model: divide span 20@10.0”**

- $E = 10.0 \times 10^6$ psi
- $L = 200$ inches
- $I = 2/3$ in$^4$
- $A = 2$ in$^2$
- $P_0 = 0.1$ lbf/in$^3$
- $g = 386.4$ in/sec$^2$

The DYNRE2 run utilizes all 19 modes calculated by STAAD (Steady State analysis) for which a value of $1.0(10)^{-10}$ times critical damping is assigned. A single forcing frequency equal to 7.5 Hz is specified for the distributed load. This load is distributed to the nodes by calculating the total integrated load for each beam and lumping one-half of this force to the respective i and j nodes.

$$F_i = F_j = \frac{x_j}{x_i} P(x) \frac{d}{dx} x = \frac{1}{2} \frac{x_j}{x_i} \frac{4}{l^2} \left(x_1 - x^2\right) dx$$
Theoretical Solution

The theoretical solution for this example is taken from:


The natural frequencies of the system are calculated using the equations from reference 1 page 108 and reference 2 page 85.

\[ f_i = \frac{\lambda_i^2}{2\pi l^2} \sqrt{\frac{EI}{m}} \]

where

\[ m = \rho A / g \]

\[ f_i = \lambda_i^2 \frac{1}{2\pi(200)^2} \sqrt{\frac{1.0(10)^6(2 / 3)}{0.10(2.0 / 386.4)}} \]

= 4.5156271*10^{-1}\lambda_i^2

\( \lambda_i \) satisfies the characteristic equation:

\[ \cos \lambda_i \cosh \lambda_i - 1 = 0 \]

Table 496: Calculated natural frequency for each mode

<table>
<thead>
<tr>
<th>i</th>
<th>( \lambda_i )</th>
<th>( \omega_i )</th>
<th>( f_i )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.730041</td>
<td>63.47865</td>
<td>10.10294</td>
</tr>
<tr>
<td>2</td>
<td>7.853205</td>
<td>174.9814</td>
<td>27.84915</td>
</tr>
<tr>
<td>3</td>
<td>10.99561</td>
<td>343.0334</td>
<td>54.59546</td>
</tr>
<tr>
<td>4</td>
<td>14.13717</td>
<td>567.0517</td>
<td>90.24907</td>
</tr>
<tr>
<td>5</td>
<td>17.27876</td>
<td>847.0773</td>
<td>134.8165</td>
</tr>
<tr>
<td>6</td>
<td>20.42035</td>
<td>1183.108</td>
<td>188.2975</td>
</tr>
<tr>
<td>7</td>
<td>23.56194</td>
<td>1575.144</td>
<td>250.6919</td>
</tr>
<tr>
<td>8</td>
<td>26.70354</td>
<td>2023.185</td>
<td>321.9998</td>
</tr>
</tbody>
</table>

The mode shapes are:

\[ \phi_i = \cosh \frac{\lambda_i}{l} - \cos \frac{\lambda_i}{l} - \sigma_i \left( \sinh \frac{\lambda_i}{l} - \sin \frac{\lambda_i}{l} \right) \]

Where:
\[ \sigma_i = \frac{\cosh \lambda_i - \cos \lambda_i}{\sinh \lambda_i - \sin \lambda_i} \]

The response of mode i to a harmonic force:
\[ \eta_i = \frac{\int \phi_i(x) P(x) \, dx}{\omega_i^2 \{ \phi_i(x) \}^2 \, dx} - \frac{\sin(\omega t - \psi_i)}{\sqrt{1 - \frac{\omega^2}{\omega_i^2} + \left( \frac{\kappa \omega}{\pi} \right)^2}} \]

Where \( \psi_i \) is the response phase lag relative to the applied force and c is the damping.

Since \( c = 0.0 \), \( \psi_i = 0.0 \)

Upon substitution and rearranging terms:
\[ \eta_i = \frac{\int l \phi_i(x) P(x) \, dx - \frac{1}{\omega_i} \int \phi_i(x)^2 \, dx \sin \omega t}{\frac{\pi}{l^2} \left( \frac{\omega_i}{\omega^2} - \omega^2 \right) \int \phi_i(x)^2 \, dx} \]

From reference 1, page 466, case c and page 467, case 29:
\[ \frac{4P_o}{\lambda_i^2} \int x \phi_i(x) \, dx - \frac{4P_o}{l^2} \int x^2 \phi_i(x) \, dx = \frac{8P_0}{\lambda_i^2 \beta_i} \left[ 1 + \left( -i \right)^i \right] - \frac{8(-i)^i P_o}{\lambda_i^2 \beta_i^2} \left[ 2 - \sigma_i \beta_i \right] \]

Since the load is symmetric, this expression is zero for \( I = 2, 4, 6 \ldots \); Therefore, \( i = 1, 3, 5 \ldots \) and: \( (-1)^i = -1 \)

And from the reference, \( \beta_i = \frac{\lambda_i}{l} \)

So:
\[ \int \phi_i(x) P(x) \, dx = \frac{16P_o l}{\lambda_i^2} \]

From Reference 1, page 457 case 5:
\[ \int \phi_i(x)^2 \, dx = 1 \]

Therefore:
\[ \omega = 7.5(2\pi) = 47.1239 \text{ radians/sec} \]

\[ \eta_i = \frac{16P_o}{m\lambda_i^2 \left( \omega_i^2 - \omega^2 \right)} \sin \omega t = \frac{16(-1.0) \sin \omega t}{\frac{386.4}{\lambda_i^2 \left( \omega_i^2 - 2 \right) \cdot 220.661^2}} = \frac{16(1.0) \sin \omega t}{\lambda_i^2 \left( \omega_i^2 - 2 \right) \cdot 220.661} \]

Table 497: Mode shapes

<table>
<thead>
<tr>
<th>( i )</th>
<th>( \lambda_i )</th>
<th>( \omega_i )</th>
<th>( \eta_i(t) )</th>
<th>( \sigma_i )</th>
<th>( \phi(1/4) )</th>
<th>( \phi(1/2) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.730041</td>
<td>63.47865</td>
<td>(-7.63815(10)^{-1}\sin \omega t)</td>
<td>0.9825022</td>
<td>0.8631319</td>
<td>1.5881463</td>
</tr>
<tr>
<td>3</td>
<td>10.99561</td>
<td>343.0334</td>
<td>(-2.21457(10)^{-3}\sin \omega t)</td>
<td>0.9999664</td>
<td>1.3708047</td>
<td>-1.4059984</td>
</tr>
</tbody>
</table>
\[ y(x, t) = \sum_{i=1,3,5,7} \eta_i(t) \phi_i(x) \]

Table 498: Steady state displacements for 1/4 point and 1/2 point (nodes 6 and 11, respectively)

<table>
<thead>
<tr>
<th>i</th>
<th>( \lambda_i )</th>
<th>( \omega_i )</th>
<th>( \eta_i(t) )</th>
<th>( \sigma_i )</th>
<th>( \phi(1/4) )</th>
<th>( \phi(1/2) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>17.27876</td>
<td>847.0773</td>
<td>(-1.44744(10)^{-4} \sin \omega t)</td>
<td>0.9999999</td>
<td>-0.5278897</td>
<td>1.4145675</td>
</tr>
<tr>
<td>7</td>
<td>23.56194</td>
<td>1575.144</td>
<td>(-2.24623(10)^{-5} \sin \omega t)</td>
<td>1.0000000</td>
<td>-1.3037973</td>
<td>-1.4141982</td>
</tr>
</tbody>
</table>

Comparison

Table 499: Comparison of results

<table>
<thead>
<tr>
<th>Location</th>
<th>Theory</th>
<th>STAAD Advanced Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node 6 (X = 50 inches)</td>
<td>0.66220</td>
<td>0.65963</td>
</tr>
<tr>
<td>Node 11 (X = 100 inches)</td>
<td>1.21011</td>
<td>1.20545</td>
</tr>
</tbody>
</table>

Steady-state analysis requires the STAAD.Pro Advanced Analysis Plus license.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Steady State Loading on a Beam.STD is included in the STAAD.Pro installation folder.

**STAAD SPACE**

START JOB INFORMATION

ENGINEER DATE 29-Mar-06

END JOB INFORMATION

* FIXED BEAM SUBJECTED TO A HARMONIC LOAD WITH A PARABOLIC DISTRIBUTION X
  * NUMBER OF NODES  21  X
  * HIGH NODE NUMBER  21  X
  * NODES FULLY RESTRAINED  2  X
  * NUMBER OF BEAM ELEMENTS  20  X
  * NUMBER OF EIGENVECTORS  19

STAAD.Pro 3634 User Manual
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 100 0; 3 200 0; 4 300 0; 5 400 0; 6 500 0; 7 600 0;
8 700 0; 9 800 0; 10 900 0; 11 1000 0; 12 1100 0; 13 1200 0;
14 1300 0; 15 1400 0; 16 1500 0; 17 1600 0; 18 1700 0; 19 1800 0;
20 1900 0; 21 2000 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 2 AY 0 AZ 0 IX 0.001 IY 0.666667 IZ 0.166667
SUPPORTS
2 TO 20 FIXED BUT FY MZ
1 21 FIXED
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
DENSITY 0.0999194
END DEFINE MATERIAL
CONSTANTS
BETA 90 ALL
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 7
CUT OFF FREQUENCY 500
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
MODAL CALCULATION REQUESTED
PERFORM STEADY STATE ANALYSIS
BEGIN STEADY FORCE
STEADY FORCE FREQ 7.5 DAMP 1e-10
JOINT LOAD
2 FY 1.8666
3 FY 3.5666
4 FY 5.0666
5 FY 6.3666
6 FY 7.4666
7 FY 8.3666
8 FY 9.0666
9 FY 9.5666
10 FY 9.8666
11 FY 9.9666
12 FY 9.8666
13 FY 9.5666
14 FY 9.0666
15 FY 8.3666
16 FY 7.4666
17 FY 6.3666
18 FY 5.0666
19 FY 3.5666
20 FY 1.8666
END
```
PRINT JOINT DISPLACEMENTS LIST 6 11
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
</tr>
</tbody>
</table>

******************************************************************************
*                              STAAD.Pro CONNECT Edition                *
*                     Version 22.01.00.**                               *
* Proprietary Program of       *
* Bentley Systems, Inc.         *
* Date= APR 14, 2019            *
* Time= 23: 6:17                *
*                               *
* Licensed to: Bentley Systems Inc *
******************************************************************************

1. STAAD SPACE

INPUT FILE: Steady State Loading on a Beam.STD

2. START JOB INFORMATION

3. ENGINEER DATE 29-MAR-06

4. END JOB INFORMATION

5. * FIXED BEAM SUBJECTED TO A HARMONIC LOAD WITH A PARABOLIC DISTRIBUTION X

6. * NUMBER OF NODES

21

7. * HIGH NODE NUMBER

21

8. * NODES FULLY RESTRAINED

2

9. * NUMBER OF BEAM ELEMENTS

20

10. * NUMBER OF EIGENVECTORS 19

11. SET SHEAR

12. UNIT INCHES POUND

13. JOINT COORDINATES

14. 1 0 0 0; 2 10 0 0; 3 20 0 0; 4 30 0 0; 5 40 0 0; 6 50 0 0; 7 60 0 0
15. 8 70 0 0; 9 80 0 0; 10 90 0 0; 11 100 0 0; 12 110 0 0; 13 120 0 0
16. 14 130 0 0; 15 140 0 0; 16 150 0 0; 17 160 0 0; 18 170 0 0; 19 180 0 0
17. 20 190 0 0; 21 200 0 0

18. MEMBER INCIDENCES

19. 1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10
20. 10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17
21. 17 17 18; 18 18 19; 19 19 20; 20 20 21

22. MEMBER PROPERTY AMERICAN

23. 1 TO 20 PRIS AX 2 AY 0 AZ 0 IX 0.001 IY 0.666667 IZ 0.166667

24. SUPPORTS

25. 2 TO 20 FIXED BUT FY MZ

26. 1 21 FIXED

27. DEFINE MATERIAL START

28. ISOTROPIC MATERIAL1

29. E 1E+07

30. POISSON 0.3

31. DENSITY 0.0999194

32. END DEFINE MATERIAL
```
33. CONSTANTS
34. BETA 90 ALL
35. MATERIAL MATERIAL1 ALL
36. CUT OFF MODE SHAPE 7
37. CUT OFF FREQUENCY 500
38. LOAD 1

STAAD SPACE  -- PAGE NO.

39. SELFWEIGHT X 1
40. SELFWEIGHT Y 1
41. SELFWEIGHT Z 1
42. MODAL CALCULATION REQUESTED
43. PERFORM STEADY STATE ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS         21  NUMBER OF MEMBERS      20
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     21

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      38
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
***NOTE: MASSES DEFINED UNDER LOAD#       1 WILL FORM
THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.

EIGEN METHOD   : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED              =     7
NUMBER OF EXISTING MASSES IN THE MODEL =    19
NUMBER OF MODES THAT WILL BE USED      =     7
***  EIGENSOLUTION : ADVANCED METHOD ***

STAAD SPACE  -- PAGE NO.

CALCULATED FREQUENCIES FOR LOAD CASE       1

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (CYCLES/SEC)</th>
<th>PERIOD (SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10.103</td>
<td>0.09898</td>
</tr>
<tr>
<td>2</td>
<td>27.849</td>
<td>0.03591</td>
</tr>
<tr>
<td>3</td>
<td>54.591</td>
<td>0.01832</td>
</tr>
<tr>
<td>4</td>
<td>90.230</td>
<td>0.01108</td>
</tr>
<tr>
<td>5</td>
<td>134.747</td>
<td>0.00742</td>
</tr>
<tr>
<td>6</td>
<td>188.093</td>
<td>0.00532</td>
</tr>
<tr>
<td>7</td>
<td>250.166</td>
<td>0.00400</td>
</tr>
</tbody>
</table>

MODAL WEIGHT (MODAL MASS TIMES g) IN POUN

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>2.759074E+01</td>
<td>0.000000E+00</td>
<td>1.584634E+01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.000000E+00</td>
<td>1.241570E-21</td>
<td>0.000000E+00</td>
<td>1.763389E+01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.000000E+00</td>
<td>5.287477E+00</td>
<td>0.000000E+00</td>
<td>1.757978E+01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.000000E+00</td>
<td>5.783502E-23</td>
<td>0.000000E+00</td>
<td>1.788088E+01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.000000E+00</td>
<td>2.138447E+00</td>
<td>0.000000E+00</td>
<td>1.901368E+01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.000000E+00</td>
<td>1.058583E-19</td>
<td>0.000000E+00</td>
<td>1.837854E+01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.000000E+00</td>
<td>1.145139E+00</td>
<td>0.000000E+00</td>
<td>1.756698E+01</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

MASS PARTICIPATION FACTORS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>72.67</td>
<td>0.00</td>
<td>0.00</td>
<td>72.666</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>72.666</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>13.93</td>
<td>0.00</td>
<td>0.00</td>
<td>86.591</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>86.591</td>
<td>0.00</td>
</tr>
</tbody>
</table>
44. BEGIN STEADY FORCE
45. STEADY FORCE FREQ 7.5 DAMP 1E-10
46. JOINT LOAD
      STAAD SPACE

47. 2 FY 1.8666
48. 3 FY 3.5666
49. 4 FY 5.0666
50. 5 FY 6.3666
51. 6 FY 7.4666
52. 7 FY 8.3666
53. 8 FY 9.0666
54. 9 FY 9.5666
55. 10 FY 9.8666
56. 11 FY 9.9666
57. 12 FY 9.8666
58. 13 FY 9.5666
59. 14 FY 9.0666
60. 15 FY 8.3666
61. 16 FY 7.4666
62. 17 FY 6.3666
63. 18 FY 5.0666
64. 19 FY 3.5666
65. 20 FY 1.8666
66. END

*DIRCTIONS FOR WHICH AMPLITUDE VS. FREQUENCY DATA WAS ENTERED = 0 2
0 0 0 0
*DIRCTIONS FOR WHICH AMPLITUDE VS. PHASE LAG DATA WAS ENTERED = 0 0
0 0 0 0

FORCE DIRECTION NUMBER 2
FREQUENCY   AMPLITUDE   PHASE ANGLE
1 0.749800E+01 0.100000E+01 0.000000E+00
2 0.750200E+01 0.100000E+01 0.000000E+00

7 MODES (EIGENVECTORS) HAVE BEEN SELECTED.

<table>
<thead>
<tr>
<th>MODE</th>
<th>DAMPED FREQUENCY</th>
<th>NATURAL FREQUENCY</th>
<th>GENERALIZED WEIGHT</th>
<th>DAMPENING COEFFICIENT</th>
<th>DAMPING COEFFICIENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(HZ)</td>
<td>(RAD/SEC)</td>
<td>(WEIGHT)</td>
<td>(MASS)</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>1.010292E+01</td>
<td>6.347852E+01</td>
<td>1.584634E+01</td>
<td>4.104327E-02</td>
<td>1.000000E-10</td>
</tr>
<tr>
<td>2</td>
<td>2.784865E+01</td>
<td>1.749782E+02</td>
<td>1.763389E+01</td>
<td>4.567317E-02</td>
<td>1.000000E-10</td>
</tr>
<tr>
<td>3</td>
<td>5.459144E+01</td>
<td>3.430081E+02</td>
<td>1.757978E+01</td>
<td>4.553302E-02</td>
<td>1.000000E-10</td>
</tr>
<tr>
<td>4</td>
<td>9.022966E+01</td>
<td>5.669297E+02</td>
<td>1.788088E+01</td>
<td>4.631289E-02</td>
<td>1.000000E-10</td>
</tr>
<tr>
<td>5</td>
<td>1.347470E+02</td>
<td>8.466405E+02</td>
<td>1.901368E+01</td>
<td>4.924693E-02</td>
<td>1.000000E-10</td>
</tr>
<tr>
<td>6</td>
<td>1.880927E+02</td>
<td>1.181821E+03</td>
<td>1.837854E+01</td>
<td>4.760188E-02</td>
<td>1.000000E-10</td>
</tr>
<tr>
<td>7</td>
<td>2.501660E+02</td>
<td>1.571839E+03</td>
<td>1.756698E+01</td>
<td>4.549988E-02</td>
<td>1.000000E-10</td>
</tr>
</tbody>
</table>

PARTICIPATION FACTORS FOR EACH MODE
<table>
<thead>
<tr>
<th>MODE NO.</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>MX</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00000E+00</td>
<td>0.218577E+04</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>2</td>
<td>0.00000E+00</td>
<td>-0.139214E-07</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>3</td>
<td>0.00000E+00</td>
<td>0.382113E+03</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>4</td>
<td>0.00000E+00</td>
<td>-0.416926E-08</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>5</td>
<td>0.00000E+00</td>
<td>-0.148269E+03</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>6</td>
<td>0.00000E+00</td>
<td>0.128546E-06</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>7</td>
<td>0.00000E+00</td>
<td>-0.829074E+02</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
<td>0.00000E+00</td>
</tr>
</tbody>
</table>

67. PRINT JOINT DISPLACEMENTS LIST 6 11
JOINT DISPLACE LIST 6

---

6. JOIN DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT LOAD X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 1 0.00000 0.65963 0.00000 0.00000 0.00000 0.01831</td>
</tr>
<tr>
<td>11 1 0.00000 1.20545 0.00000 0.00000 0.00000 0.00000</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

68. FINISH

******** END OF THE STAAD.Pro RUN ********
**** DATE= APR 14,2019  TIME= 23: 6:18 ****

---

V.09 Steel Design
V. Australia
V. Steel Design per AS 4100-1998
V. AS4100 1998 - Bending Capacity

To check the bending capacity of the UB shape per AS4100-1998.

Reference

Problem
Check the suitability of a 360UB50.7 Grade 300 for the beam in Example 5.2 of the reference. This solution relies mainly on AS 4100 and minimally on AISC [1999a], for example, to get basic section properties. Omit deflection checks.

Comparison
Table 500: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending capacity, Mnz (kN·m)</td>
<td>138</td>
<td>137.5</td>
<td>0.3%</td>
</tr>
<tr>
<td>Bending moment (kN·m)</td>
<td>125</td>
<td>124.7</td>
<td>0.2%</td>
</tr>
<tr>
<td>Utilization ratio</td>
<td>0.906</td>
<td>0.907</td>
<td>0.1%</td>
</tr>
</tbody>
</table>

The reference book does not compute the ratio.

STAAD Input

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNEC...S\Verification Models\09 Steel Design\Australia\AS4100 1998\AS4100 1998 - Bending Capacity.STD is typically installed with the program.

STAAD PLANE BENDING CAPACITY PER AS4100 1998

* OBJECTIVE : TO DETERMINE THE ADEQUACY OF A UB SHAPE IN BENDING PER THE AS4100-1998 CODE
* START JOB INFORMATION
ENGINEER DATE 22-Jul-14
END JOB INFORMATION
* INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 8 0 0;
*
MEMBER INCIDENCES
1 1 2;
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST UB360X50.7
PRINT MEMBER PROPERTIES ALL
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 2 PINNED
*****************************************************************
LOAD 1 LOADTYPE DEAD TITLE PERMANENT LOADS - G
MEMBER LOAD
* UB Self weight
1 UNI GY -0.54
* Slab
1 UNI GY -3.328
*
LOAD 2 LOADTYPE Live TITLE IMPOSED LOADS - Q
MEMBER LOAD
* Imposed Action
1 UNI GY -4.8
* Point Load
1 CON GY -10
*****************************************************************
LOAD COMBINATION 3 ULC
1 1.2 2 1.5
*****************************************************************
PERFORM ANALYSIS
*
UNIT MMS NEWTON
PARAMETER 1
CODE AUSTRALIAN 1998
*
* Grade 300 steel
FYLD 300 ALL
*
* Twist restraint factor
KT 1.04 ALL
*
* Effective length Le
UNT 5820 ALL
*
* Moment modification factor
ALM 1.41 ALL
* TRACK 2 ALL
* CHECK CODE ALL
* Published capacity:- Mb = 138 (pg 114)
* Therefore Utilisation ratio = 125/138 = 0.906
* FINISH

### STAAD Output

STAAD.Pro CODE CHECKING - ( AS4100-1998 )

V2.3

NOTE : BY DEFAULT "AS/NZS 3679.1 300" WILL BE USED FOR ROLLED SECTIONS.

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>FORCE/MOMENT</th>
<th>IN NEWTON MM</th>
<th>IN KN METRE</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>207.7</td>
<td>+</td>
<td>PNC=0.2405E+3</td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>54.0</td>
<td>+</td>
<td>PNT=0.1747E+4</td>
</tr>
<tr>
<td>UNL =</td>
<td>5820.0</td>
<td></td>
<td>0.0000E+0</td>
</tr>
<tr>
<td>MAIN =</td>
<td>0.0</td>
<td></td>
<td>0.0000E+0</td>
</tr>
<tr>
<td>PHI =</td>
<td>0.90</td>
<td></td>
<td>0.0000E+0</td>
</tr>
<tr>
<td>FULT =</td>
<td>440.0</td>
<td></td>
<td>0.0000E+0</td>
</tr>
<tr>
<td>FYLD =</td>
<td>300.0</td>
<td></td>
<td>0.0000E+0</td>
</tr>
<tr>
<td>NSF =</td>
<td>1.00</td>
<td>+</td>
<td>VZ =0.6371E-3</td>
</tr>
<tr>
<td>SKT =</td>
<td>1.00 -6.9</td>
<td></td>
<td>0.0000E+0</td>
</tr>
<tr>
<td>SKL =</td>
<td>1.00</td>
<td></td>
<td>0.0000E+0</td>
</tr>
<tr>
<td>SKR =</td>
<td>1.00</td>
<td></td>
<td>0.0000E+0</td>
</tr>
</tbody>
</table>

Section Type: Compact - about Z axis; Compact - about Y axis

Parameters used to calculate RATIO

Ns=0.1723E+4  Msz=0.2691E+3  Msy=0.5053E+2  Mbz=0.1528E+3
Miz=0.1130E-4  Miy=0.1130E-4  Moz=0.1130E-4
Nciz=0.1433E+4  Ncicy=0.2672E+3  Ncz=0.1433E+4  Ncy=0.2672E+3
Vvmz=0.7079E+3  Vvmz=0.7079E+3

MAX FORCE/ MOMENT SUMMARY ( KN-METR)

<table>
<thead>
<tr>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOMENT-Y</th>
<th>MOMENT-Z</th>
</tr>
</thead>
</table>

Verification Examples
V.09 Steel Design
STAAD.Pro 3642 User Manual
V. AS4100 1998 - Combined Compression and Bending Capacity

To check the capacity of a member in axial compression plus bending per AS4100-1998.

Reference

Problem
Verify the capacity of the beam-column which uses a UC section in Grade 300 steel. The column is fixed at the base and braced at mid-point in weak-axis bending only. The top of the column is pinned and laterally restrained by braces. Beam B1 and B2 are connected to the column using simple construction to AS 4100 Clause 4.3.4.

Permanent Loads:
- From Beam B1: Py = 260 kN, Mz = -52 kN·m
- From Beam B2: Py = 78.1 kN, Mx = 6.248 kN·m

Imposed Loads:
- From Beam B1: Py = 190 kN, Mz = -38 kN·m
- From Beam B2: Py = 60 kN, Mx = 41.8 kN·m

Comparison

Table 501: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Load N* (kN)</td>
<td>791</td>
<td>790.9</td>
<td>&lt; 0.1%</td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.09 Steel Design

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major Moment, Mx* (kN·m)</td>
<td>119</td>
<td>119.4</td>
<td>0.3%</td>
</tr>
<tr>
<td>Minor Moment, My* (kN·m)</td>
<td>14.7</td>
<td>14.7</td>
<td>none</td>
</tr>
<tr>
<td>Axial Capacity, 0.9×Ns (kN)</td>
<td>2,870</td>
<td>0.9(3,192.4) = 2,873.2</td>
<td>0.1%</td>
</tr>
<tr>
<td>Major Capacity, 0.9×Msx (kN)</td>
<td>310</td>
<td>0.9(344.4) = 310.0</td>
<td>none</td>
</tr>
<tr>
<td>Minor Capacity, 0.9×Msy (kN)</td>
<td>143</td>
<td>0.9(158.8) = 142.9</td>
<td>&lt;0.1%</td>
</tr>
<tr>
<td>Ratio (per Clause 8.3.4)</td>
<td>0.762</td>
<td>0.763</td>
<td>0.1%</td>
</tr>
</tbody>
</table>

#### STAAD Input

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998\AS4100 1998 - Combined Compression and Bending Capacity.STD is typically installed with the program.

STAAD SPACE COMPRESSION+BENDING PER AS4100 1998


OBJECTIVE: TO DETERMINE THE ADEQUACY OF A UC SHAPE IN COMPRESSION PLUS BENDING PER THE AS4100-1998 CODE

START JOB INFORMATION
ENGINEER DATE 22-Jul-14
END JOB INFORMATION

* INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 9 0;
MEMBER INCIDENCES
1 1 2;
* DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST UC250X89.5
PRINT MEMBER PROPERTIES ALL
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 FIXED
2 FIXED BUT FY MX MZ
**********************************
LOAD 1 LOADTYPE Dead TITLE PERMANENT LOADS
JOINT LOAD
* Beam 1 RG1
  2 FY -260 MZ -52
* Beam 2 RG2
  2 FY -78.1 MX 6.248
* Column Self Weight
  2 FY -8.5
*
LOAD 2 LOADTYPE Live TITLE IMPOSED LOADS
JOINT LOAD
* Beam 1 RG1
  2 FY -190 MZ -38
* Beam 2 RG2
  2 FY -60 MX 4.8
*
LOAD COMB 3 ULS
1 1.2 2 1.5
*
PERFORM ANALYSIS
*
UNIT MMS NEWTON
PARAMETER 1
CODE AUSTRALIAN 1998
*
* Grade 300 steel
FYLD 280 ALL
*
* Restrained mid height in minor axis
LZ 9000 ALL
LY 4500 ALL
UNT 4500 ALL
UNB 4500 ALL
*
* Effective length factor Kex
KZ 0.85 ALL
*
* Effective Length factor Key
KY 1 ALL
*
TRACK 2 ALL
CHECK CODE ALL
*
* Published ratio for clause 8.3.4 = 0.762
*
FINISH
**STAAD Output**

* STAAD.Pro CODE CHECKING - ( AS4100-1998 )

V2.3

*******************************************************************************

NOTE : BY DEFAULT "AS/NZS 3679.1 300" WILL BE USED FOR ROLLED SECTIONS.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>DESIGN CODE</th>
<th>Y PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>AS4100 1998</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>AUSTRALIAN SECTIONS</th>
<th>IN CM UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST UC250X89.5</td>
<td></td>
</tr>
</tbody>
</table>

| ------------ | ========= | ----------- |
| LENGTH ME= 9.00 | RY=0.6516E+1 | RZ=0.1120E+2 |

<table>
<thead>
<tr>
<th>FORCE/MOMENT</th>
<th>IN NEWTON MM</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>FORCE/MOMENT</td>
</tr>
</tbody>
</table>

| KL/R-Y= 69.1 | PNC=0.2091E+4 |
| KL/R-Z= 68.3 | PNT=0.2873E+4 |
| UNL = 4500.0 | p=0.7909E+3   |
| MAIN = 0.0   | MNZ=0.3100E+3 |
| PHI = 0.90   | mnz=0.1194E+3 |
| FULT = 440.0 | MNY=0.1429E+3 |
| FYLD = 280.0 | VZ =0.1339E+4 |
| NSF = 1.00   | VV =0.3578E+3 |
| SKT = 1.00   | vV =0.2448E+1 |
| SKL = 1.00   | SKR = 1.00   |

| ----- | ========= | ----------- |
| SECTION TYPE | (WITH LOAD NO.) |
| Compact - about Z axis; | Absolute MZ Envelope |

| Ns=0.3192E+4 | Msz=0.3444E+3 |
| Msy=0.1588E+3 | Mbz=0.3444E+3 |
| Miz=0.2151E+3 | Mly=0.9875E+2 |
| Moz=0.2142E+3 |
| Nciz=0.2341E+4 | Nciy=0.2324E+4 |
| Nciz=0.2341E+4 | Ncy=0.2324E+4 |
| Vvmy=0.3976E+3 |
| Vvmz=0.1488E+4 |
| ALPHA,M= 2.475 |
| ALPHA,B= 0.000 |
| ALPHA,SZ= 0.824 |

MAX FORCE/ MOMENT SUMMARY ( KN-METR)

<table>
<thead>
<tr>
<th>790.9</th>
<th>19.8</th>
<th>2.4</th>
<th>14.7</th>
<th>119.4</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>3</td>
</tr>
</tbody>
</table>

DESIGN SUMMARY ( KN-METR)

PASS AS-8.4.4.1 0.619 3
V. AS4100 1998 - Single Angle Section in Tension

To check the tensile capacity of a single angle per AS4100-1998.

Reference

Problem
Select a section for a diagonal member of a truss. Verify its capacity. Use on equal leg angle in Grade 300 steel with one line of M20/S fasteners in one leg. Axial load of 300 kN is applied.

Comparison

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Applied tensile load (kN)</td>
<td>300</td>
<td>300.3</td>
<td>0.1%</td>
</tr>
<tr>
<td>Tensile capacity, 0.9×Nt (kN)</td>
<td>492</td>
<td>492</td>
<td>none</td>
</tr>
<tr>
<td>Ratio</td>
<td>0.610</td>
<td>0.610</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

* Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998\AS4100 1998 - Single Angle Section in Tension.STD is typically installed with the program.

STAAD SPACE AXIAL TENSILE CAPACITY PER AS4100 1998


* OBJECTIVE : TO DETERMINE THE CAPACITY OF A SINGLE ANGLE IN TENSION

* PER THE AS4100-1998 CODE

* INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+008
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-006
DAMP 0.03
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST A125X125X8
PRINT MEMBER PROPERTY ALL
*
CONSTANTS
MATERIAL STEEL MEMB 1
*
SUPPORTS
1 FIXED
*********************************************************************
LOAD 1 LOADTYPE DEAD TITLE PERMANENT LOAD
JOINT LOAD
2 FX 99.0
*
LOAD 2 LOADTYPE LIVE TITLE IMPOSED LOAD
JOINT LOAD
2 FX 121
*
LOAD COMBINATION 3 ULS
1 1.2 2 1.5
*
PERFORM ANALYSIS
UNIT MMS NEWTON
*
PARAMETER 1
CODE AUSTRALIAN 1998
*
* KT from AS 4100-1998 Table 7.3.2 case (i)
KT 0.85 MEMB 1
*
* NSF = An/Ag = 1720/1900
NSF 0.905 MEMB 1
***
TRACK 2.0 ALL
CHECK CODE MEMB 1
*
* UTILIZATION RATIO = 300/492 = 0.610
* 
FINISH
**STAAD Output**

* STAAD.Pro CODE CHECKING - ( AS4100-1998 )

**V2.3**

******************************************************************************
NOTE : BY DEFAULT "AS/NZS 3679.1 300" WILL BE USED FOR ROLLED SECTIONS.

<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER</td>
</tr>
<tr>
<td>-------------</td>
</tr>
<tr>
<td>MEMBER 1</td>
</tr>
<tr>
<td>DESIGN CODE</td>
</tr>
</tbody>
</table>

**PARAMETER**

- **KL/R-Y=** 81.1
- **KL/R-Z=** 161.3
- **UNL =** 4000.0
- **MAIN =** 0.0
- **FULT =** 440.0
- **FYLD =** 320.0
- **NSF =** 0.90
- **SKT =** 1.00
- **SKL =** 1.00
- **SKR =** 1.00

**Section Type:** Noncompact - about Z axis; Noncompact - about Y axis

Parameters used to calculate RATIO

<table>
<thead>
<tr>
<th>Ns</th>
<th>Msz</th>
<th>Msz</th>
<th>Msz</th>
<th>Msz</th>
<th>Moz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Msz</td>
<td>0.9458E+1</td>
<td>0.2831E+2</td>
<td>0.7607E+1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Msz</td>
<td>0.0000E+0</td>
<td>0.0000E+0</td>
<td>0.0000E+0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nciz</td>
<td>0.1111E+3</td>
<td>0.2725E+3</td>
<td>0.1111E+3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nciz</td>
<td>0.0000E+0</td>
<td>0.0000E+0</td>
<td>0.0000E+0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**MAX FORCE/ MOMENT SUMMARY ( KN-METR)**

<table>
<thead>
<tr>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOMENT-Y</th>
<th>MOMENT-Z</th>
</tr>
</thead>
</table>

Verification Examples

V.09 Steel Design

STAAD.Pro 3649 User Manual
V. Canadian

V. CAN/CSA S16-01

In the next few pages are included several verification examples for reference purposes. Since the S16-01 code is similar in many respects to the previous edition of the code (CAN/CSA S16.1-94), the solved examples of the 1994 edition of the CISC Handbook have been used as reference material for these examples.

V. CSA S16-01 - Cantilever with Biaxial Loading

A slender, cantilever beam subjected to a uniform load. Static analysis, 3D beam element.

Reference


Problem

A cantilever beam of length 4 meter is subjected to uniformly distributed load of 3 KN/Meter in both major and minor axis. Axial compression of 8 KN is also applied to the member. User defined steel section Sect_Class-4 from is assigned to the member.

Given

Design forces

- 8.0 KN (Compression)
- 6.0 KNm (Bending-Y)
- 6.0 KNm (Bending-Z)
- 6.0 KN (Shear-Y)
- 6.0 KN (Shear-Z)

Section Properties(Sect_Class-4):
Area = 2766 mm²
Depth of section, D = 150 mm
Thickness of web Tw = 7 mm
Width of flange Bf = 150 mm
Thickness of flange Tf = 6 mm
Moment of inertia about Z axis, Iz = 1086.96 × 10⁴ mm⁴
Moment of inertia about Y axis, Iy = 337.894 × 10⁴ mm⁴
Moment of inertia about X axis, Ix = 3.7378 × 10⁴ mm⁴
Warping constant, Cw = 1.752 × 10¹⁰ mm⁶

Member Length L = 2 m, Unbraced length = 100 mm.

Material
FYLD = 300 MPa
E = 2.05E+05 MPa
G = E/2.6 MPa

Solution

Slenderness Ratio
Effective Length factor along Local Y-Axis = KY = 1
Effective Length factor along Local Z-Axis = KZ = 1
Slenderness ratio about Z axis, L/Rz = 31.9
Slenderness ratio about Y axis, L/Ry = 57.22
Maximum Slenderness Ratio, L/Rmax = 57.22

Section Classification
Bf/Tf = 150×0.5/6 = 12.5 > 200/sqrt(Fy) = 11.54
Flange is Class 4.
d/Tw = (150-2.0×6)/7 = 19.714
(1100/sqrt(Fy))*(1-0.39*Cf/ϕ*Cy)=(1100/sqrt(300))*(1-0.39*8000/(0.9*2766*300)) = 63.24
Web is Class 1.
Overall section is Class 4 section.

Check against axial compression (Clause 13.3.3)
Effective width, Beff = 200*Tf/sqrt(300) = 69.24
Effective area, Aeff = 69.24×6×4+(150-2×6)×7 = 2627.76 mm⁴.
Effective yield stress, FYLDeff =40000/(0.5*Bf/Tf)⁴ = 256 MPa.
As per Clause 13.3.3(a),
Elastic critical buckling, Fe = π⁴×E/L,Rmax⁴ = 617.956 MPa.
Non-dimensional slenderness ratio, $\lambda = \sqrt{\frac{FYLD}{Fe}} = 0.697$

Axial compressive resistance, $Cr = \phi \cdot A_{eff} \cdot FYLD \cdot (1+0.697^{2 \cdot 1.34})^{-1/1.34} = 557886.104 \text{ N.}$

As per Clause 13.3.3(b),

Elastic critical buckling, $Fe = \frac{\pi^4 \cdot E}{L_{Rmax}^4} = 617.956 \text{ MPa.}$

Effective non-dimensional slenderness ratio, $\lambda_{eff} = \sqrt{\frac{FYLD_{eff}}{Fe}} = 0.644$

Axial compressive resistance, $Cr = \phi \cdot A_{eff} \cdot FYLD_{eff} \cdot (1+0.644^{2 \cdot 1.34})^{-1/1.34} = 521726.94 \text{ N.}$

Axial compressive resistance $\text{Min}(557886.104, 521726.94) = 521726.94 \text{ N.}$

Check against bending (Clause 13.5(c))

As the web of the section meets the requirement of Class 3 and flange exceeds Class 3 limit, flexural resistance should be calculated as per clause 13.5(c).iii.

Effective moment of inertia about Z axis,

$$I_{zeff} = \frac{2 \cdot (2 \cdot 69.24 \cdot 6^3)}{12} + \frac{2 \cdot (2 \cdot 69.24 \cdot 6) \cdot (150-6) \cdot (150-6) / 4 + (7 \cdot (150-2 \cdot 6)^3) / 12}{10152591.12 \text{ mm}^4}. $$

Effective section modulus about Z axis,

$$S_{zeff} = 10152591.12 \cdot 2 / 150 = 135367.88 \text{ mm}^3.$$  

Effective moment of inertia about Y axis,

$$I_{yeff} = \frac{(2 \cdot 6 \cdot (2 \cdot 69.24)^3)}{12} + \frac{(0.5 \cdot (150-6)^3)}{12} = 2657648.856 \text{ mm}^4. $$

Effective section modulus about Y axis,

$$S_{yeff} = 2657648.856 / 69.24 = 38383.144 \text{ mm}^3.$$  

Major axis bending resistance if member is laterally supported,

$$Mrz1 = \phi \cdot S_{zeff} \cdot FYLD = 0.9 \cdot 135367.88 \cdot 300 = 36549327.6 \text{ N-mm.} $$

Minor axis bending resistance,

$$Mry = \phi \cdot S_{yeff} \cdot FYLD = 0.9 \cdot 38383.144 \cdot 300 = 10363448.88 \text{ N-mm.} $$

If the member is laterally unsupported major axis bending resistance is determined by clause 13.6(b).

As the value of one of the end moments is 0.0, $\omega_2 = 1.75.$

Where, as per clause 13.6(a),

$$Mu = (1.75 \cdot 3.14 / 2000) \cdot \sqrt{(205000 \cdot 337.894 \cdot 10^{14} \cdot 78846.154 \cdot 3.7378 \cdot 10^4 + (3.14 \cdot 205000 / 2000) \cdot 337.894 \cdot 10^{14} \cdot 1.752 \cdot 10^9)} \cdot 2.48 \cdot 10^8 $$

$$My = Sz \cdot FYLD = (1086.96 \cdot 10^4 \cdot X2 / 150) \cdot 300 = 43478400. $$

Since $Mu > 0.65My,$

Moment of resistance $Mrz2 = 1.15 \cdot 0.9 \cdot 43478400 \cdot (1 - 0.28 \cdot 43478400 / 2.48 \cdot 10^8) = 42791153.71 \text{ N-mm = 42.79 KN-m.}$

$Mrz2$ should not be more than $Mrz1.$ Since, $Mrz2 > Mrz1$ in this example, $Mrz2 = Mrz1.$

$Mrz2 = 36549327.6 \text{ N-mm = 36.549 KN-m}$
Comparison

Table 503: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Hand Calculation</th>
<th>STAAD.Pro Result</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial compressive resistance (kN)</td>
<td>521.73</td>
<td>521.9</td>
<td>none</td>
</tr>
<tr>
<td>Major axis bending resistance (kN-m)</td>
<td>36.549</td>
<td>36.57</td>
<td>none</td>
</tr>
<tr>
<td>Minor axis bending resistance (kN-m)</td>
<td>10.363</td>
<td>10.38</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2001\CSA S16-01 - Cantilever with Biaxial Loading.STD is typically installed with the program.

STAAD SPACE VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91
START JOB INFORMATION
ENGINEER DATE 16-FEB-10
END JOB INFORMATION
* CISC EXAMPLE 1 PAGE 5-91, LIMIT STATES DESIGN, CSA-S16.1-94
* SIMPLY SUPPORTED BEAM WITH UNIFORM LOAD
* LIVE LOAD DEFLECTION OF L/300
UNIT MMS KN
JOINT COORDINATES
1 0 0 0; 2 2000 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT METER NEWTON
WIDE FLANGE
SECT_CLASS-4
0.002766 0.15 0.007 0.15 0.006 1.08696e-05 3.37894e-06 3.7378e-08 -
0.00105 0.0018
END
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.05e+08
POISSON 0.3
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 SECT_CLASS-4
UNIT MMS KN
STAAD Output

The design output from STAAD.Pro:

```
STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01 )
V2.1
******************************************************************************
ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/  CRITICAL COND/  RATIO/  LOADING/
FIELD   MY             MZ       LOCATION
=======================================================================
1 ST   SECT_CLASS-4             (UPT)                  PASS     CSA-13.8.3B        0.760         1
8.00 C         -6.00           6.00        0.00
VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91        -- PAGE NO.
5
ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/  CRITICAL COND/  RATIO/  LOADING/
FIELD   MY             MZ       LOCATION
=======================================================================

MEMBER PROPERTIES (UNIT = CM)
--------------------------------
CROSS SECTION AREA =  2.77E+01   MEMBER LENGTH =  2.00E+02
IZ =  1.09E+03   SZ =  1.45E+02   PZ =  1.63E+02
IY =  3.38E+02   SY =  4.51E+01   PY =  6.92E+01
IX =  3.74E+00   CW =  1.75E+04
EFFECTIVE MEMBER PROPERTIES FOR CLASS-4 SECTION(UNIT = CM)
--------------------------------
EFFECTIVE CROSS SECTION AREA =  2.63E+01
EFFECTIVE IZ =  1.02E+03   EFFECTIVE SZ =  1.35E+02
EFFECTIVE IY =  2.66E+02   EFFECTIVE SY =  3.85E+01
EFFECTIVE YIELD STRESS = 256.0  MPA
COMPRESSIVE CAPACITIES FOR CLASS 4 SECTION(UNIT = MPA)
--------------------------------
BASED ON EFFECTIVE AREA
CR1 =  7.098E+02   CR2 =  5.582E+02   CRZ =  6.705E+02
```
V. CSA S16-01 - Wide Flange Beam Interaction Ratio 1

Reference


Problem

Find the interaction ratio, beam resistance and beam deflection.

Given

\[ E = 200000 \text{ MPa (STEEL)} \]
\[ F_y = 300 \text{ Mpa CSA G40.21-M} \]

Simply supported beam has a 8.0 m span; Ky is 1.0, Kz 1.0, unsupported length 1.0 m

Allowable Live Load deflection, L/300 = 8000/300 = 27 mm

Factored Uniform Load IS 7 kN/m DEAD, 15 kN/m LIVE.

Steel section is W410X54
**Comparison**

**Table 504: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interaction Ratio</td>
<td>0.88</td>
<td>0.882</td>
<td>negligible</td>
</tr>
<tr>
<td>Beam Resistance (kN·m)</td>
<td>284</td>
<td>283.5</td>
<td>negligible</td>
</tr>
<tr>
<td>Beam Deflection (mm)</td>
<td>21</td>
<td>21.39</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2001\CSA S16-01 - Wide Flange Beam Interaction Ratio 1.STD` is typically installed with the program.

```
STAAD SPACE VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91
START JOB INFORMATION
ENGINEER DATE 21-Sep-18
END JOB INFORMATION
  * CISC EXAMPLE 1 PAGE 5-91, LIMIT STATES DESIGN, CSA-S16.1-94
  * SIMPLY SUPPORTED BEAM WITH UNIFORM LOAD
  * LIVE LOAD DEFLECTION OF L/300
UNIT MMS KN
JOINT COORDINATES
  1 0 0 0; 2 8000 0 0;
MEMBER INCIDENCES
  1 1 2;
MEMBER PROPERTY CANADIAN
    1 TABLE ST W410X54
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
  E 1.99947e+08
  POISSON 0.3
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
  1 PINNED
  2 FIXED BUT MY MZ
UNIT METER KN
LOAD 1 DEAD
MEMBER LOAD
  1 UNI GY -7
LOAD 2 LIVE
MEMBER LOAD
  1 UNI GY -15
LOAD COMB 3 1.25DL + 1.5 LL
  1 1.25 2 1.5
PERFORM ANALYSIS
LOAD LIST 2
PRINT SECTION DISPL
```
### STAAD Output

#### MEMBER SECTION DISPLACEMENTS

---

**UNITS ARE - CM**

<table>
<thead>
<tr>
<th>MEMB</th>
<th>LOAD</th>
<th>GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>0.0000 0.0000 0.0000 0.0000 -0.5623 0.0000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0000 -1.0822 0.0000 0.0000 -1.5237 0.0000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0000 -1.8590 0.0000 0.0000 -2.0681 0.0000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0000 -2.1392 0.0000 0.0000 -2.0681 0.0000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0000 -1.8590 0.0000 0.0000 -1.5237 0.0000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0000 -1.0822 0.0000 0.0000 -0.5623 0.0000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0000 0.0000 0.0000 0.0000 0.0000 0.0000</td>
</tr>
</tbody>
</table>

**MAX LOCAL DISP = 2.13916 AT 400.00 LOAD 2 L/DISP= 373**

************ END OF SECT DISPL RESULTS ************
V. CSA S16-01 - Wide Flange Beam Interaction Ratio 2

Reference


Problem

Find the interaction ratio, beam and column resistance.

Given

\( E = 200,000 \) MPa (STEEL).
\( F_y = 300 \) MPa   CSA G40.21-M

Simply supported beam/column has a 3.7 m span, Ky is 1.0, Kz 1.0. Factored axial load is 2000 kN and end moments of 200 kN·m and 300 kN·m.

Steel section is W310X118
Comparison

Table 505: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interaction Ratio</td>
<td>0.940</td>
<td>0.941</td>
<td>negligible</td>
</tr>
<tr>
<td>Beam Resistance (kN-m)</td>
<td>606</td>
<td>605.5</td>
<td>negligible</td>
</tr>
<tr>
<td>Column Resistance (kN)</td>
<td>3,850</td>
<td>3,849</td>
<td>negligible</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2001\CSA S16-01 - Wide Flange Beam Interaction Ratio 2.STD is typically installed with the program.

STAAD SPACE VERIFICATION CISC HANDBOOK 9th Edition PAGE 4-114
START JOB INFORMATION
ENGINEER DATE 14-Nov-18
JOB NAME Verification Example S16-01 : Compression + Major Bending
JOB CLIENT Bentley Systems Inc
JOB NO CISC
JOB REV 1.0
JOB PART 1
END JOB INFORMATION
*
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.7 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X118
*
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
*
*
UNIT MMS NEWTON
DEFINE MATERIAL START
ISOTROPIC STEEL_ASTMA992
E 200000
POISSON 0.3
DENSITY 7.68191e-05
ALPHA 6.5e-06
DAMP 0.03
G 76902.7
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
```plaintext
END DEFINE MATERIAL
*
CONSTANTS
MATERIAL STEEL_ASTMA992 ALL
*
*
UNIT METER KN
LOAD 1 AXIAL
JOINT LOAD
2 FY -2000
LOAD 2 BENDING MAJOR
JOINT LOAD
2 MZ 200
1 MZ 300
LOAD 3 LOADTYPE None TITLE AXIAL+BENDINGMAJOR
REPEAT LOAD
1 1.0 2 1.0
*
PERFORM ANALYSIS
*
LOAD LIST 3
*
*
UNIT MMS NEWTON
PARAMETER 1
CODE CANADIAN 2001
TRACK 2 ALL
LY 3700 ALL
LZ 3700 ALL
CMZ 0.4 ALL
FYLD 345 ALL
CHECK CODE ALL
*
STEEL MEMBER TAKE OFF ALL
FINISH

SAAD Output

STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01 )
V2.1

*******************************************************
ALL UNITS ARE - KNS MET (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
======================================================
1 ST W310X118 (CANADIAN SECTIONS)
PASS CSA-13.8.2C 0.941 3
2000.00 C 0.00 300.00 0.00
VERIFICATION CISC HANDBOOK 9TH EDITION PAGE 4-114 -- PAGE NO.
5
ALL UNITS ARE - KNS MET (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
======================================================
MEMBER PROPERTIES (UNIT = CM)
-----------------------------
CROSS SECTION AREA = 1.50E+02 MEMBER LENGTH = 3.70E+02

```
IZ = 2.75E+04   SZ = 1.75E+03   PZ = 1.95E+03
IY = 9.02E+03   SY = 5.88E+02   PY = 8.93E+02
IX = 1.60E+02   CW = 1.97E+06
MATERIAL PROPERTIES (UNIT = MPA)
--------------------------------
FYLD = 345.0   FU = 396.7   E = 2.00E+05   G = 7.69E+04
SECTION CAPACITIES (UNIT - KN,M)
---------------------------------
CR1 = 4.658E+03  CR2 = 3.849E+03  SECTION CLASS 2
CRZ = 4.443E+03  CTORFLX = 3.849E+03
TENSILE CAPACITY = 4.553E+03  COMPRRESSIVE CAPACITY = 3.849E+03
FACTORED MOMENT RESISTANCE : MRY = 2.773E+02  MRZ = 6.055E+02
          MU = 2.300E+03
FACTORED SHEAR RESISTANCE : VRY = 7.657E+02  VRZ = 1.571E+03
MISCELLANEOUS INFORMATION
--------------------------
NET SECTION FACTOR FOR TENSION = 1.000
KL/RY = 47.714   KL/RZ = 27.326   ALLOWABLE KL/R = 200.000
UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) = 3.700
OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 0.40   OMEGA-2 = 1.00
SHEAR FORCE (KNS) : Y AXIS = 1.351E+02   Z AXIS = 0.000E+00
SLENDERNESS RATIO OF WEB (H/W) = 2.32E+01

V. CSA S16-01 - Wide Flange Beam Interaction Ratio 3

Reference


Problem

Find the interaction ratio, beam, and column resistance.

Given

E = 200,000 MPa (STEEL)
Fy = 345 MPa (ASTM A992)

Simply supported beam/column has a 3.7 m span, Ky is 1.0, Kz 1.0, Lu = 3.7 m

The factored axial load is 2,000 kN and end moments of 200 kN·m and 300 kN·m in the strong axis and 50 kN·m at each end in the weak axis.

Steel section is W310X129.
### Comparison

Table 506: CAN/CSA-S16 comparison

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interaction Ratio</td>
<td>0.990</td>
<td>0.990</td>
<td>none</td>
</tr>
<tr>
<td>Beam Resistance, Weak axis (kN·m)</td>
<td>308</td>
<td>307.7</td>
<td>negligible</td>
</tr>
<tr>
<td>Beam Resistance, Strong axis (kN·m)</td>
<td>671</td>
<td>670.7</td>
<td>negligible</td>
</tr>
<tr>
<td>Column Resistance (kN)</td>
<td>4,240</td>
<td>4,242</td>
<td>negligible</td>
</tr>
</tbody>
</table>

### STAAD Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\9 Steel Design\Canada\S16 2001\CSA S16-01 - Wide Flange Beam Interaction Ratio 3.STD is typically installed with the program.

STAAD SPACE VERIFICATION CISC HANDBOOK 9th Edition PAGE 4-115
START JOB INFORMATION
ENGINEER DATE 14-Nov-18
JOB NAME Verification Example S16-01 : Compression + Bending(s)
JOB CLIENT Bentley Systems Inc
JOB NO CISC
JOB REV 1.0
JOB PART 1
END JOB INFORMATION
*
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.7 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X129
*
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
*
*
UNIT MMS NEWTON
DEFINE MATERIAL START
ISOTROPIC STEEL_ASTM_A992
E 2000000
POISSON 0.3
DENSITY 7.68191e-05
ALPHA 6.5e-06
DAMP 0.03
G 76982.7
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
*
CONSTANTS
MATERIAL STEEL_ASTM_A992 ALL
*
*
UNIT METER KN
LOAD 1 AXIAL
JOINT LOAD
2 FY -2000
LOAD 2 BENDING MAJOR
JOINT LOAD
2 MZ 200
1 MZ 300
LOAD 3 BENDING MINOR
JOINT LOAD
2 MX 50
1 MX 50
LOAD 4 LOADTYPE None TITLE AXIAL+BENDING(S)
REPEAT LOAD
1 1.0 2 1.0 3 1.0
*
PERFORM ANALYSIS
*
LOAD LIST 4
*
*
UNIT MMS NEWTON
PARAMETER 1
CODE CANADIAN 2001
TRACK 2 ALL
LY 3700 ALL
LZ 3700 ALL
SSY 0 ALL
SSZ 0 ALL
FYLD 345 ALL
CHECK CODE ALL
*
STEEL MEMBER TAKE OFF ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01 )
V2.1
*****************************************************************************
ALL UNITS ARE - KNS MET (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST W310X129 (CANADIAN SECTIONS) PASS CSA-13.8.2B 0.990 4
2000.00 C -50.00 300.00 0.00
V. CAN/CSA S16-09

V.CSA S16-09 - Axial Tension

Design of a double-angle member subject to tension load.

Reference

Problem
Design a tension diagonal in an all-welded truss (SNU 0 for welded). Attached to WT 265x61.5 chords (t_w = 13.1 mm).

L = 4 m

T_f = 630 kN

The material used is G40.21 300W steel (F_y = 300 MPa, F_u = 450 MPa) (FYLD and FU)
Calculations

Gross area is used for welded connections.

\[ T_r = \max \left\{ \phi AF_y = 0.9 \times 300 \times A = 270A, \phi_u AF_u = 0.75 \times 450 \times A = 338A \right\} \]

\[ T_r \geq T_f, \text{ so solve for } A: \]

\[ A = \frac{630 \times (10)^3}{270} = 2,333 \text{ mm}^2 \]

Select a pair of angles with at least this gross area: try a 2-76x64x9.5, with long legs back-to-back:

\[ A = 2,480 \text{ mm}^2 \]
\[ r_{\min} = 13.3 \text{ mm} \]

Calculate the effective net area due to shear lag. The average weld length is taken as \((120 \text{ mm} + 250 \text{ mm})/2 = 185 \text{ mm}\).

The long leg does not have a reduction in area shear lag, thus:

\[ A_{n2} = 1.00 \times (76 - 9.5) \times 9.5 = 632 \text{ mm}^2 \]

The outstanding (short) leg has an eccentricity of half the leg length, or \(64/2 = 32 \text{ mm}\).

\[ A_{n3} = \left(1 - \frac{32}{250}\right) \times 64 \times 9.5 = 530 \text{ mm}^2 \]

The total net area for the pair of angles for the rupture limit state is:

\[ A_{ne} = 632 + 530 = 1,162 \text{ mm}^2 \]

For the pair of angles, \(A_{ne} = 2 \times 1,162 = 2,324 \text{ mm}^2\); the ratio of net section area to gross area = \(2,324 / 2,480 = 0.937 (\text{NSF})\).

Checking the yielding limit state:

\[ T_r = 0.90 \times 2,480 \times 300 = 670 \text{ kN} \]

Checking the rupture limit state:

\[ T_r = 0.75 \times 2,324 \times 450 = 784 \text{ kN} \]

Yielding governs, and the critical ratio is: \(630/670 = 0.94\).

Comparison

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(T_r) (kN)</td>
<td>670</td>
<td>669.6</td>
<td>negligible</td>
</tr>
<tr>
<td>Critical Ratio</td>
<td>0.94</td>
<td>0.941</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**Note:** If the member SELECT facility is used in place of a code CHECK, STAAD.Pro will actually select a 2-L 102x89x6.4 member, which has a critical ratio of 0.997 (per Cl. 13.2) and an area of 2,340 mm\(^2\), thus more economical (the NSF used for this size is 0.923).
STAAD.Pro Input File
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Axial Tension.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE LD L76X64X9.5
*1 TABLE T W530x123
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FX 630
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 300000 ALL
FU 450000 ALL
SNUG 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD.Pro Output

STAAD.PRO CODE CHECKING - S16-09 (v1.1)
***********************************************************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
--------------------------------- START OF DESIGN OUTPUT OF MEMBER 1
### Verification Examples

#### V.09 Steel Design

<table>
<thead>
<tr>
<th>MEMBER NO: 1</th>
<th>CRITICAL RATIO: 0.941(PASS)</th>
<th>LOAD: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION (MET): 0.00</td>
<td>CONDITION: Cl. 13.9.1</td>
<td></td>
</tr>
<tr>
<td>SECTION: LD L76X64X9.5 (CANADIAN SECTIONS)</td>
<td>UNIT: KN MET</td>
<td></td>
</tr>
<tr>
<td>STRENGTH CHECKS:</td>
<td>CRITICAL RATIO: 0.941(PASS)</td>
<td>LOAD CASE: 1</td>
</tr>
<tr>
<td>DESIGN FORCES:</td>
<td>Fx: 630.00(T)</td>
<td>Fy: 0.00</td>
</tr>
<tr>
<td>Mx: 0.00E+00</td>
<td>My: 0.00E+00</td>
<td>Mz: 0.00E+00</td>
</tr>
<tr>
<td>SECTION PROPERTIES:</td>
<td>AZZ: 11.195</td>
<td>AYY: 13.616</td>
</tr>
<tr>
<td>MATERIAL PROPERTIES:</td>
<td>FYLD: 300.000</td>
<td>FU: 450.000</td>
</tr>
<tr>
<td>ACTUAL MEMBER LENGTH (MET): 4.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PARAMETERS:</td>
<td>KZ: 1.000</td>
<td>KY: 1.000</td>
</tr>
<tr>
<td>SLENDERNESS:</td>
<td>ACTUAL SLENDERNESS RATIO: 169.796</td>
<td>LOAD: 1</td>
</tr>
<tr>
<td>ALLOWABLE SLENDERNESS RATIO: 300.000</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Section Class:
- COMPRESSION: Class 1
- FLEXURE: Class 3
- FLANGE: Class 3
- UNIT: KN MET

#### Tension:
- FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
  - MAJOR: 0.000 141.791 0.000 Cl. 13.3
  - MINOR: 0.000 166.729 0.000 Cl. 13.3

#### Intermediate:
- Ag:(CM) KL/r: Fe:(N MM) l n:
  - MAJOR: 24.800 169.796 70.178 2.068 1.340
  - MINOR: 24.800 154.539 84.719 1.882 1.340

#### Intermediate:
- FLEX TORS BUCK:
  - FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
    - 0.000 164.708 0.000 Cl. 13.3 1

#### Intermediate:
- SHEAR:
  - FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
    - 0.000 226.444 0.000 Cl. 13.4.3

#### Section Class:
- TENSION: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
  - MAJOR: 0.000 196.070 0.000 AISC G2-1
<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>INTERMEDIATE:</td>
<td>Aw: (CM)</td>
<td>Kv:</td>
<td>Ka:</td>
<td>Fcri: (N MM)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fs: (N MM)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Major:</td>
<td>13.616</td>
<td>1.200</td>
<td>0.000</td>
<td>0.000</td>
<td>11.195</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Minor:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>UNIT: KN MET</td>
<td>FORCE:</td>
<td>CAPACITY:</td>
<td>RATIO:</td>
<td>CRITERIA:</td>
<td>LOAD CASE:</td>
<td></td>
</tr>
<tr>
<td>Location (MET):</td>
<td>Major:</td>
<td>0.00E+00</td>
<td>7.18E+00</td>
<td>0.000</td>
<td>Cl. 13.5 (b)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Minor:</td>
<td>0.00E+00</td>
<td>7.08E+00</td>
<td>0.000</td>
<td>Cl. 13.5 (b)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.000</td>
<td>INTERMEDIATE:</td>
<td>Mp:</td>
<td>Se:</td>
<td>My:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Major:</td>
<td>1.44E+01</td>
<td>0.00E+00</td>
<td>0.80E+01</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Minor:</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.79E+01</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>UNIT: KN MET</td>
<td>Lat Tor Buck:</td>
<td>Force:</td>
<td>Capacity:</td>
<td>Ratio:</td>
</tr>
<tr>
<td>Location (MET):</td>
<td>Major:</td>
<td>0.00E+00</td>
<td>1.17E+01</td>
<td>0.000</td>
<td>Cl. 13.6 (e) (i)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Minor:</td>
<td>0.00E+00</td>
<td>2.50</td>
<td>1.00</td>
<td>0.59</td>
<td>3.65E+01</td>
</tr>
<tr>
<td></td>
<td></td>
<td>UNIT: KN MET</td>
<td>Interaction:</td>
<td>Ratio:</td>
<td>Criteria:</td>
<td>Load Case:</td>
</tr>
<tr>
<td>Location (MET):</td>
<td>FLEXURE AND AXIAL TENSION:</td>
<td>C/S STRENGTH:</td>
<td>0.941</td>
<td>Cl. 13.9.1</td>
<td>1</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>FLEXURE AND AXIAL COMPRESSION:</td>
<td>C/S STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.3</td>
<td>1</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>MEMBER STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.3</td>
<td>1</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>LTB STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.3</td>
<td>1</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FLEX AND SHEAR:</td>
<td>0.000</td>
<td>Cl. 14.6(a)</td>
<td>1</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>BIAXIAL FLEX:</td>
<td>0.000</td>
<td>Cl. 13.8</td>
<td>1</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>STAAD SPACE</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>INTERMEDIATE:</td>
<td>FLEXURE AND AXIAL TENSION:</td>
<td>C/S STRENGTH:</td>
<td>Tf:</td>
<td>6.30E+02</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FLEXURE AND AXIAL COMPRESSION:</td>
<td>Cf:</td>
<td>Mfz:</td>
<td>Mrz:</td>
<td>Mry:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C/S STR:</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>6.70E+02</td>
</tr>
<tr>
<td></td>
<td></td>
<td>+02</td>
<td>2.11E+02</td>
<td>0.60</td>
<td>0.60</td>
<td>1.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>MEM STR:</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>1.42E+02</td>
</tr>
<tr>
<td></td>
<td></td>
<td>+02</td>
<td>2.11E+02</td>
<td>0.60</td>
<td>0.60</td>
<td>1.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>LTB STR:</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>1.65E+02</td>
</tr>
<tr>
<td></td>
<td></td>
<td>+02</td>
<td>2.11E+02</td>
<td>0.60</td>
<td>0.60</td>
<td>1.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FLEX SHEAR:</td>
<td>Mfz:</td>
<td>0.00E+00</td>
<td>Mrz:</td>
<td>7.18E+00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>+02</td>
<td>BIAX FLEX:</td>
<td>Mfz:</td>
<td>0.00E+00</td>
<td>Mrz:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>+00</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
V.CSA S16-09 - Beam Bending

Determine the uniformly factored load that the member can resist.

Reference

Problem
A W310x52 beam spans 7.3 m. Both ends of the beam are supported by columns connected standard web angle connections.
The material used is G40.21 350W steel (\(F_y = 350\) MPa) (FYLD)

Calculations
The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Section Classification
Evaluate the slenderness effects of the beam flanges:
\[
\frac{b}{2t} = \frac{167}{2 \times 13.2} = 6.3 < \frac{170}{\sqrt{350}} = 9.1
\]
Evaluate the slenderness effects of the beam web:
\[
\frac{h}{w} = \frac{318 - 2(13.2)}{7.6} = 38.4 < \frac{1}{700} \frac{700}{\sqrt{350}} = 90.9
\]
Both the beam flanges and web are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 5.7.

Bending Capacity
Factored moment resistance is:
\[
M_r = \phi Z_x F_y = 0.90 \times 841 \times (10)^3 \times 350 = 265 \times (10)^3 \text{N} \cdot \text{mm} = 265 \text{kN} \cdot \text{m}
\]
Equate this to the bending moment due to a uniformly distributed load and solve for the load:
\[
w_f \geq \frac{8M_r}{L^2} = \frac{8 \times 265}{(7.3)^2} = 39.8 \text{ kN/m}
\]
### Comparison

**Table 508: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_r$ (kN·m)</td>
<td>265</td>
<td>265</td>
<td>none</td>
</tr>
<tr>
<td>$w_f$ (kN/m)</td>
<td>39.8</td>
<td>39.76*</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**Note:** (*) STAAD.Pro does calculate the allowable uniform load on the beam. Instead, this is done by applying a uniform load incrementally until the critical ratio is ≈ 1.0. The, this uniform load is divided by the resulting critical ratio to normalize the distributed load capacity of the beam. Thus, $39.8/1.001 = 39.67$ kN/m

---

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Beam Bending.STD is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X52
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -39.8
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
```
STAAD.Pro Output

STAAD.PRO CODE CHECKING - S16-09 (v1.1)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)

START OF DESIGN OUTPUT OF MEMBER 1

*MEMBER NO: 1  CRITICAL RATIO: 1.001(FAIL) LOAD: 1
LOCATION (MET): 3.65 CONDITION: Cl. 13.8
SECTION: ST W310X52 (CANADIAN SECTIONS)
UNIT: KN MET

STRENGTH CHECKS:
CRITICAL RATIO: 1.001(FAIL) LOAD CASE: 1 LOCATION (MET): 3.65
CONDITION: Cl. 13.8
DESIGN FORCES: Fx: 0.00(T) Fy: 0.00 Fz: 0.00
Mx: 0.00E+00 My: 0.00E+00 Mz: -2.65E+02
UNIT: CM

SECTION PROPERTIES: AZZ: 44.088 AYY: 23.165 CW: 237980.875
SZZ: 748.428 SYY: 123.353 IZZ: 11900.002 IYY: 1030.000
UNIT: NEW MM

MATERIAL PROPERTIES: FYLD: 350.000 FU: 450.000
ACTUAL MEMBER LENGTH(MET): 7.300
PARAMETERS: KZ: 1.000 KY: 1.000 NSF: 1.000 SLF: 1.000

SLENDERNESS: ACTUAL SLENDERNESS RATIO: 185.766 LOAD: 1 LOC.
(MET): 0.000

SECTION CLASS:
COMPRESSION: Class 4
FLEXURE: Class 1
FLANGE: Class 1
WEB: Class 1
UNIT: KN MET

TENSION:
FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:

LOCATION(MET):
YIELDING: 0.000 2101.050 0.000 Cl. 13.2
1 0.000

RUPTURE: 0.000 2251.125 0.000 Cl. 13.2
1 0.000

STAAD SPACE

STAAD.PRO CODE CHECKING - S16-09 (v1.1)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)

COMPRESSION:
FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:

LOCATION(MET):
MAJOR: 0.000 1587.513 0.000 Cl. 13.3.5(a)
1 0.000
MINOR:           0.000     322.454      0.000    Cl. 13.3.5(a)
1
INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)         l           n:
MAJOR:          65.224     54.653      677.375      0.719        1.340
MINOR:          65.224    185.766       58.630      2.443        1.340
FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):  0.000    1023.539      0.000    Cl. 13.3.5(a)      1
0.000
INTERMEDIATE:   Fe:(N MM)  l             n:
SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):  -145.270     502.453      0.289    Cl. 13.4.1.1
1
MINOR:           0.000     833.263      0.000    AISC G2-1
1
Fs:(N MM)
MAJOR:        -23.165    231.000      231.000      0.044       326.992
MINOR:          0.000
UNIT: KN MET
YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):  2.65E+02    2.65E+02      1.001    Cl. 13.5(a)
1
MINOR:        0.00E+00    5.95E+01      0.000    Cl. 13.5(a)
1
INTERMEDIATE:   Mp:        Se:         My:
MAJOR:        2.94E+02    5.95E+01    0.00E+00
MINOR:        6.62E+01    5.95E+01    0.00E+00
UNIT: KN MET
INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
FLEXURE AND AXIAL TENSION:
C/S STRENGTH:               1.001    Cl. 13.9.1         1       3.650
FLEXURE AND AXIAL COMPRESSION:
C/S STRENGTH:               0.851    Cl. 13.8.2         1       3.650
MEMBER STRENGTH:            0.851    Cl. 13.8.2         1       0.000
FLEX AND SHEAR:             0.729    Cl. 14.6(a)        1       3.042
BIAXIAL FLEX:              1.001    Cl. 13.8
INTERMEDIATE:
FLEXURE AND AXIAL TENSION:
C/S STRENGTH:               0.00E+00  2.65E+02      2.65E+02    2.65E+02 Mrz:
2.65E+02
STAAD SPACE
-- PAGE NO.
7
STAAD.PRO CODE CHECKING - (S16-09) v1.1
*****************************************************************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
FLEXURE AND AXIAL COMPRESSION:
Cf:               Mfz:    Mfy:    Cr:    Mrz:    Mry:    b : 
Cez:  Cey:    w 1z:  w 1y: U1z: U1y:
C/S STR: 0.00E+00  2.65E+02  0.00E+00  2.05E+03  2.65E+02  5.95E+01  0.85 4.52E+03
   3.91E+02  0.60 0.60 1.00 1.00
MEM STR: 0.00E+00  2.65E+02  0.00E+00  3.22E+02  2.65E+02  5.95E+01  0.85 4.52E+03
   3.91E+02  0.60 0.60 1.00 1.00
V.CSA S16-09 - Beam Shear Capacity

Reference

Problem
The W530x82 beam has a maximum shear force, \( v = 540 \text{ kN} \) due to factored loads.
\( L = 11.0 \text{ m} \)

The material used is G40.21 350W steel (\( F_y = 350 \text{ MPa} \)) (FYLD)

Calculations
Evaluate the slenderness effects of the beam web:
\[
\frac{h}{w} = \frac{528 - 2(13.3)}{9.5} = 52.8 < \frac{439}{5.34} \frac{F_y}{F_y} = 439 \sqrt{\frac{5.34}{350}} = 54.2
\]

Therefore the shear capacity is calculated as:
\[
V_r = \phi \times A_w \times F_y = 0.9 \times 528 \times 9.5 \times 350 = 1,043 \text{ kN}
\]

Calculate Equivalent Concentrated Load
The given end moments and assumed distributed loads (which was determined to given an equivalent mid-span moment of the given example) do not result in a shear force of 540 kN. Therefore, an additional concentrated load is added near the first support.
\[
R_1 = \frac{w_f L}{2} + \frac{M_1 - M_2}{L} = \frac{40.17 \times 11.0}{2} + \frac{540 - 185}{11.0} = 253.2 \text{ kN}
\]

\[
P = 540 \text{ kN} \times 253.2 \text{ kN} = 287 \text{ kN}
\]

The STAAD.Pro will have this load applied at 1 mm away from the left support.
Comparison

Table 509: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(V_r) (kN)</td>
<td>1,043</td>
<td>1,042.826</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Beam Shear Capacity.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 ENFORCED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
1 MZ 540
2 MZ -185
MEMBER LOAD
1 UNI GY -40.033
1 CON GY -352.27 10.945
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
STAAD.Pro Output

1

******************************************************************************
* *
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 23: 7:27 *
* *
* Licensed to: Bentley Systems Inc *
******************************************************************************

1. STAAD SPACE

INPUT FILE: CSA S16-09 - Beam Shear Capacity.STD
2. START JOB INFORMATION
3. ENGINEER DATE 17-MAR-19
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER KN
7. JOINT COORDINATES
8. 1 0 0 0; 2 11 0 0
9. MEMBER INCIDENCES
10. 1 1 2
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 2.05E+08
14. POISSON 0.3
15. DENSITY 76.92
16. ALPHA 1.2E-05
17. DAMP 0.03
18. TYPE STEEL
19. STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
20. G 7.692E+07
21. END DEFINE MATERIAL
22. MEMBER PROPERTY CANADIAN
23. 1 TABLE ST W530X82
24. CONSTANTS
25. MATERIAL STEEL ALL
26. SUPPORTS
27. 1 PINNED
28. 2 ENFORCED BUT FX MY MZ
29. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
30. JOINT LOAD
31. 1 MZ 540
32. 2 MZ -185
33. MEMBER LOAD
34. 1 UNI GY -40.033
35. 1 CON GY -352.27 10.945
36. PERFORM ANALYSIS

STAAD SPACE
**Problem Statistics**

- Number of Joints: 2
- Number of Members: 1
- Number of Plates: 0
- Number of Solids: 0
- Number of Surfaces: 0
- Number of Supports: 2

Using 64-bit analysis engine.

**SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER**

Total Primary Load Cases = 1, Total Degrees of Freedom = 9
Total Load Combination Cases = 0 so far.

37. Print Member Properties All

**STAAD Space** -- Page No.

3

**Member Properties. Unit - CM**

<table>
<thead>
<tr>
<th>MEMB</th>
<th>PROFILE</th>
<th>AX/</th>
<th>IZ/</th>
<th>IY/</th>
<th>IX/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ST W530X82</td>
<td>105.00</td>
<td>47700.00</td>
<td>2030.00</td>
<td>51.80</td>
</tr>
<tr>
<td>1</td>
<td></td>
<td>50.16</td>
<td>37.06</td>
<td>1806.82</td>
<td>194.26</td>
</tr>
</tbody>
</table>

************ END OF DATA FROM INTERNAL STORAGE ************

38. Print Support Reaction All

**STAAD Space** -- Page No.

4

**Support Reactions - Unit: KN METE**

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1</td>
<td>0.00</td>
<td>254.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2 1</td>
<td>0.00</td>
<td>538.42</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

************ END OF LATEST ANALYSIS RESULT ************

39. Parameter 1
40. Code Canadian 2009
41. FYLD 350000 All
42. FU 450000 All
43. TRACK 2 All
44. CHECK CODE ALL

**Steel Design**

**STAAD Space** -- Page No.

5

**STAAD.PRO CODE CHECKING - S16-09 (v1.1)**

*All units are - KN MET (unless otherwise noted)*

--- START OF DESIGN OUTPUT OF MEMBER 1 ---

*MEMBER NO: 1 CRITICAL RATIO: 1.803(FAIL) LOAD: 1
LOCATION (MET): 0.00 CONDITION: Cl. 13.8
SECTION: ST W530X82 (CANADIAN SECTIONS)
UNIT: KN MET

**Strength Checks:**

- **Critical Ratio:** 1.803(FAIL)
- **Load Case:** 1
- **Location (MET):** 0.00

**Condition:** Cl. 13.8

**Design Forces:**

- **Fx:** 0.00(T)
- **Fy:** 254.22
- **Fz:** 0.00
- **Mx:** 0.00E+00
- **My:** 0.00E+00
- **Mz:** 5.40E+02

**UNIT:** CM

**Section Properties:**

- **AZZ:** 55.594
- **AYY:** 48.896
- **CW:** 1340255.500
- **SZZ:** 1806.818
- **SYY:** 194.258
- **IZZ:** 47700.008
- **IYY:** 2030.000
**Verification Examples**

**V.09 Steel Design**

<table>
<thead>
<tr>
<th>UNIT: NEW MM</th>
<th>MATERIAL PROPERTIES: FYLD: 350.000 FU: 450.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTUAL MEMBER LENGTH(MET): 11.000</td>
<td></td>
</tr>
<tr>
<td>PARAMETERS: KZ: 1.000 KY: 1.000 NSF: 1.000 SLF: 1.000</td>
<td></td>
</tr>
<tr>
<td>SLENDERNESS: ACTUAL SLENDERNESS RATIO: 250.172 LOAD: 1 LOC.</td>
<td></td>
</tr>
<tr>
<td>ALLOWABLE SLENDERNESS RATIO: 300.000</td>
<td></td>
</tr>
<tr>
<td>SECTION CLASS: COMPRESSION: Class 4 FLEXURE: Class 2 FLANGE: Class 2 WEB: Class 1</td>
<td></td>
</tr>
</tbody>
</table>
| UNIT: KN MET TENSION: | \begin{array}{|c|c|c|c|}
| LOCATION(MET): & FORCE: & CAPACITY: & RATIO: & CRITERIA: \end{array} |
| YIELDING: 0.000 3307.500 0.000 Cl. 13.2 |
| RUPTURE: 0.000 3543.750 0.000 Cl. 13.2 |
| -- PAGE NO. -- |
| 6 |

**STAAD SPACE**

**STAAD.PRO CODE CHECKING - (S16-09) v1.1**

**ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>LOCATION(MET):</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAJOR: 0.000</td>
<td>2253.274</td>
<td>0.000</td>
<td>Cl. 13.3.5(a)</td>
<td></td>
</tr>
<tr>
<td>MINOR: 0.000</td>
<td>253.222</td>
<td>0.000</td>
<td>Cl. 13.3.5(a)</td>
<td></td>
</tr>
<tr>
<td>INTERMEDIATE: 0.000</td>
<td>854.388</td>
<td>0.000</td>
<td>Cl. 13.3.5(a)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LOCATION(MET):</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAJOR: 48.896</td>
<td>345.654</td>
<td>0.048</td>
<td>231.000</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LOCATION(MET):</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAJOR: -5.40E+02</td>
<td>6.49E+02</td>
<td>0.832</td>
<td>345.654</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LOCATION(MET):</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAJOR: -5.40E+02</td>
<td>6.49E+02</td>
<td>0.832</td>
<td>345.654</td>
<td></td>
</tr>
</tbody>
</table>

**Verification Examples**

**V.09 Steel Design**

**STAAD.Pro 3677 User Manual**
**Verification Examples**

V.09 Steel Design
V.CSA S16-09 - Select a Beam

Select a wide-flange shape for the given beam span.

**Reference**


**Problem**

A beam must be selected to span 11.0 m. The end moment of the span are:

\[ M_1 = -540 \text{ kN m} \]
\[ M_2 = -185 \text{ kN m} \]

And the maximum moment in the span is:

\[ M_3 = 256 \text{ kN m} \]

The beam is braced at 2.5 m from the center of each supporting column (thus \( L_b = 6.0 \text{ m} \)) (LT 6.0).

The material used is G40.21 350W steel \( (F_y = 350 \text{ MPa}) \) (FYLD)

**Calculations**

Assume a Class 2 section:

\[ M_r = \phi M_p = \phi Z_x F_y \]

Thus,

\[ Z_x \geq \frac{540(10)^6}{0.9 \times 350} = 1,714 \times (10)^3 \text{ mm}^3 \]

Try a W460x82:

\[ Z_x = 1,830\times(10)^3 \text{ mm}^3, \text{ tf} = 16.0 \text{ mm}, \text{ bf} = 191.0 \text{ mm}, \text{ tw} = 9.9 \text{ mm}, \text{ D} = 460.0 \text{ mm} \]

Evaluate the slenderness effects of the beam flanges:

\[ \frac{b}{2t} = \frac{191}{2 \times 16.0} = 5.7 < \frac{170}{\sqrt{350}} = 9.1 \]

Evaluate the slenderness effects of the beam web:

\[ \frac{h}{w} = \frac{460 - 2(16.0)}{9.9} = 43.2 < \frac{1,700}{\sqrt{350}} = 90.9 \]

The assumptions are valid and this beam is sufficient to support the given moments.

\[ M_r = 0.9\times1,830\times(10)^3\times350 \times [(10)^6] = 576.5 \text{ kN m} \]
Ratio = \frac{540}{576.5} = 0.937

**Comparison**

Table 510: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section</td>
<td>W460x82 *</td>
<td>W460x82</td>
<td>none</td>
</tr>
<tr>
<td>Mₘ (kN-m)</td>
<td>576.5 **</td>
<td>576</td>
<td>negligible</td>
</tr>
<tr>
<td>Critical Ratio</td>
<td>0.937 **</td>
<td>0.937</td>
<td>none</td>
</tr>
</tbody>
</table>

**Notes:**

*The reference also tries a W530x82, which is stiffer for the same weight. STAAD.Pro selects the shallower beam.

** The reference does not calculate the resisting moment capacity nor the critical ratio for either section size, but they are evaluated here for completeness.

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Select a Beam.STD is typically installed with the program.

In order to model this beam in STAAD.Pro, we need to determine the distributed load which would result in the same mid-span moment on a beam with the given end moments. From the AISC Steel Construction Manual, p 3-222:

\[ M₃ = \frac{w_f L^2}{8} - \frac{(M₁ + M₂)}{2} + \frac{(M₁ - M₂)^2}{2w_f L^2} \]

\[ 256 = \frac{w_f (11.0)^2}{8} - \frac{(540 + 185)^2}{2} + \frac{(540 - 185)^2}{2w_f (11.0)^2} \]

Solving this yields \( w_f = 40.033 \text{ kN/m} \)

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -40.033
JOINT LOAD
1 MZ 540
2 MZ -185
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
SELECT ALL
PERFORM ANALYSIS
FINISH

**STAAD.Pro Output**

1

*******************************************************************************
* 
* STAAD.Pro CONNECT Edition 
* Version 22.01.00.** 
* Proprietary Program of 
* Bentley Systems, Inc. 
* Date= APR 14, 2019 
* Time= 23: 7:32 
* 
* Licensed to: Bentley Systems Inc 
*******************************************************************************

1. STAAD SPACE
INPUT FILE: CSA S16-09 - Select a Beam.STD
2. START JOB INFORMATION
3. ENGINEER DATE 17-MAR-19
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER KN

_VERIFICATION EXAMPLES_
7. JOINT COORDINATES
8. 1 0 0 0; 2 11 0 0
9. MEMBER INCIDENCES
10. 1 1 2
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 2.05E+08
14. POISSON 0.3
15. DENSITY 76.92
16. ALPHA 1.2E-05
17. DAMP 0.03
18. TYPE STEEL
19. STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
20. G 7.692E+07
21. END DEFINE MATERIAL
22. MEMBER PROPERTY CANADIAN
23. 1 TABLE ST W530X82
24. CONSTANTS
25. MATERIAL STEEL ALL
26. SUPPORTS
27. 1 PINNED
28. 2 FIXED BUT FX MY MZ
29. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
30. MEMBER LOAD
31. 1 UNI GY -40.033
32. JOINT LOAD
33. 1 MZ 540
34. 2 MZ -185
35. PERFORM ANALYSIS

STAAD SPACE

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       6
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

36. PRINT MEMBER PROPERTIES ALL

STAAD SPACE

MEMBER PROPERTIES. UNIT - CM
-----------------
MEMB  PROFILE              AX/          IZ/          IY/        IX/
      AY           AZ           SZ         SY
1  ST W530X82          105.00     47700.00      2030.00       51.80
     50.16        37.06      1806.82      194.26

************ END OF DATA FROM INTERNAL STORAGE ************

37. PRINT SUPPORT REACTION ALL

STAAD SPACE

SUPPORT REACTIONS -UNIT KN     METE  STRUCTURE TYPE = SPACE
------------------
JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM Z
38. PARAMETER 1
39. CODE CANADIAN 2009
40. LAT 1 ALL
41. FYLD 350000 ALL
42. FU 450000 ALL
43. TRACK 2 ALL
44. CHECK CODE ALL

STEEL DESIGN

--- PAGE NO.
5

STAAD.PRO CODE CHECKING - S16-09 (v1.1)
*******************************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
--------------------------- START OF DESIGN OUTPUT OF MEMBER 1
---------------------------

MEMBER NO: 1  CRITICAL RATIO: 0.832(PASS)  LOAD: 1
LOCATION (MET): 0.00  CONDITION: Cl. 13.8
SECTION: ST W530X82  (CANADIAN SECTIONS)
UNIT: KN MET

STRENGTH CHECKS:
CRITICAL RATIO: 0.832(PASS)  LOAD CASE: 1 LOCATION (MET): 0.00
CONDITION: Cl. 13.8

DESIGN FORCES:  
Fx: 0.00(T)   Fy: 252.45  Fz: 0.00
Mx: 0.00E+00  My: 0.00E+00  Mz: 5.40E+02
UNIT: CM

SECTION PROPERTIES:
AZZ: 55.594 AYY: 48.896 CW: 1340255.500
SZZ: 1806.818 SYY: 194.258
IZZ: 47700.008 IYY: 2030.000
UNIT: NEW MM

MATERIAL PROPERTIES:
FYLD: 350.000  FU: 450.000

ACTUAL MEMBER LENGTH(MET): 11.000
PARAMETERS:
KZ: 1.000  KY: 1.000  NSF: 1.000  SLF: 1.000

SLENDERNESS:
ACTUAL SLENDERNESS RATIO: 250.172  LOAD: 1 LOC.
(MET): 0.000
ALLOWABLE SLENDERNESS RATIO: 300.000

SECTION CLASS:
COMPRESSION: Class 4
FLEXURE: Class 2
FLANGE: Class 2
WEB: Class 1
UNIT: KN MET

TENSION:
FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
YIELDING: 0.000 3307.500 0.000 Cl. 13.2
1 0.000
RUPTURE: 0.000 3543.750 0.000 Cl. 13.2
1 0.000

--- PAGE NO.
6

STAAD.PRO CODE CHECKING - ( S16-09) v1.1
*******************************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
COMPRESSION: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
MAJOR: 0.000 2253.274 0.000 Cl. 13.3.5(a)
1 0.000
MINOR: 0.000 253.222 0.000 Cl. 13.3.5(a)
1 0.000
INTERMEDIATE: Ag:(CM) KL/r: Fe:(N MM) 1 n:
MAJOR: 89.688 51.609 759.622 0.679 1.340
MINOR: 89.688 250.172 32.328 3.290 1.340
FLEX TOR BUCK: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
0.000 854.388 0.000 Cl. 13.3.5(a) 1
INTERMEDIATE: Fe:(N MM) l n:
SHEAR: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
MAJOR: 252.454 1042.826 0.242 Cl. 13.4.1.1
1 0.000
MINOR: 0.000 1050.727 0.000 AISC G2-1
1 0.000
Fs:(N MM)
MAJOR: 48.896 5.349 0.048 237.748
345.654 231.000
MINOR: 55.594
UNIT: KN MET
YIELDING: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
MAJOR: -5.40E+02 6.49E+02 0.832 Cl. 13.5(a)
1 0.000
MINOR: 0.00E+00 9.54E+01 0.000 Cl. 13.5(a)
1 0.000
INTERMEDIATE: Mp: Se: My:
UNIT: KN MET
INTERACTION: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
FLEXURE AND AXIAL TENSION:
C/S STRENGTH: 0.832 Cl. 13.9.1 1 0.000
FLEXURE AND AXIAL COMPRESSION:
C/S STRENGTH: 0.707 Cl. 13.8.2 1 0.000
MEMBER STRENGTH: 0.707 Cl. 13.8.2 1 0.000
FLEX AND SHEAR: 0.715 Cl. 14.6(a) 1 0.000
BIAXIAL FLEX:
INTERMEDIATE:
FLEXURE AND AXIAL TENSION:
C/S STRENGTH: Tf: 0.00E+00 Tr: 3.31E+03 Mfz: -5.40E+02 Mrz: 6.49E+02
STAAD SPACE
-------- PAGE NO.
7
STAAD.PRO CODE CHECKING - ( S16-09) v1.1
******************************************************************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
FLEXURE AND AXIAL COMPRESSION:
Cf: Mfz: Mfy: Cr: Mrz: Mry: b :
Cez: Cey: w 1z: w 1y: U1z: U1y:
C/S STR: 0.00E+00 -5.40E+02 0.00E+00 2.83E+03 6.49E+02 9.54E+01 0.85 7.98E
**MEM STR:** 0.00E+00 -5.40E+02 0.00E+00 2.52E+02 6.49E+02 9.54E+01 0.85 7.98E+03 3.39E+02 0.60 0.74 1.00 1.00

**FLEX SHEAR:**  Mfz: -5.40E+02 Mrz: 6.49E+02 Vfy: 2.52E+02 Vry: 1.04E+03

**BIAX FLEX:**  Mfz: -5.40E+02 Mrz: 6.49E+02 Mfy: 0.00E+00 Mry: 9.54E+01

**---------------- END OF DESIGN OUTPUT OF MEMBER 1**

**NOTE:** OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL MEMBERS

45. SELECT ALL STEEL DESIGN

**STAAD SPACE**

--- PAGE NO. 8

**STAAD.PRO MEMBER SELECTION - S16-09 (v1.1)**

--- PAGE NO. 9

**STAAD.PRO MEMBER SELECTION - ( S16-09) v1.1**
### Compression

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Force (KN)</th>
<th>Capacity (KN)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.000</td>
<td>2261.248</td>
<td>0.000</td>
<td>Cl. 13.3.5(a)</td>
<td>1</td>
</tr>
<tr>
<td>Minor</td>
<td>0.000</td>
<td>253.374</td>
<td>0.000</td>
<td>Cl. 13.3.5(a)</td>
<td>1</td>
</tr>
</tbody>
</table>

### Interimmediate

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Force (KN)</th>
<th>Capacity (KN)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>96.728</td>
<td>58.319</td>
<td>0.767</td>
<td>1.340</td>
<td>1</td>
</tr>
<tr>
<td>Minor</td>
<td>96.728</td>
<td>260.107</td>
<td>3.421</td>
<td>1.340</td>
<td>1</td>
</tr>
</tbody>
</table>

### Flexure Bucking

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Force (KN)</th>
<th>Capacity (KN)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>1279.603</td>
<td>0.000</td>
<td>Cl. 13.3.5(a)</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

### Shear

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Force (KN)</th>
<th>Capacity (KN)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>252.454</td>
<td>946.776</td>
<td>0.267</td>
<td>Cl. 13.4.1.1</td>
<td>1</td>
</tr>
<tr>
<td>Minor</td>
<td>0.000</td>
<td>1155.168</td>
<td>0.000</td>
<td>AISC G2-1</td>
<td>1</td>
</tr>
</tbody>
</table>

### Yielding

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Force (KN)</th>
<th>Capacity (KN)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-5.40E+02</td>
<td>5.76E+02</td>
<td>0.937</td>
<td>Cl. 13.5(a)</td>
<td>1</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>9.54E+01</td>
<td>0.000</td>
<td>Cl. 13.5(a)</td>
<td>1</td>
</tr>
</tbody>
</table>

### Flexure and Axial Tension

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.937</td>
<td>Cl. 13.9.1</td>
<td>1</td>
</tr>
</tbody>
</table>

### Flexure and Axial Compression

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.796</td>
<td>Cl. 13.8.2</td>
<td>1</td>
</tr>
</tbody>
</table>

### Bi-axial Flexure

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.802</td>
<td>Cl. 14.6(a)</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**STAAD Space**

---

**Verifications Examples**

V.09 Steel Design

---

**STAAD.Pro Members Selection**

S16-09 v1.1

---

**User Manual**

---
V.CSA S16-09 - Shear Capacity Combined Stresses

Reference

Problem
The W250x73 column has the following concentrated loads

\[ C_f = 900 \text{ kN} \]

And a uniform moment of:

\[ M_{fs} = 180 \text{ kN m} \]

The material used is G40.21 350W steel \( (F_y = 350 \text{ MPa}) \) (FYLD). The beam is braced against lateral-torsional buckling only at the ends (LAT 0, default)
**Calculations**

**Axial Capacity**

Determine the buckling load of the column:

\[
\left( \frac{KL}{r} \right)_x = \frac{1.06 (3, 600)}{110} = 32.7
\]

\[
\left( \frac{KL}{r} \right)_y = \frac{1.06 (3, 600)}{64.6} = 55.7
\]

The largest slenderness ratio controls:

\[
F_{ex} = \frac{n^2 E}{KL} \left( \frac{55.7}{x} \right)^2 = 635.6 \text{ kN}
\]

\[
\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{635.6}} = 0.742
\]

\[
C_r = \phi A F_y \left( 1 + \lambda^{2n} \right)^{-1/n} = 0.90 \times 9, 280 \times 350 \left[ 1 + 0.742 \left( 2 \times 1.34 \right) \right]^{-1/1.34} = 2, 216 \times (10)^6 \text{ N} = 2, 216 \text{ kN}
\]

**Moment Modifier**

Uniform moment:

\[
\omega_1 = 1.0
\]

Calculate the elastic buckling load for the bending axis:

\[
C_e = \frac{\pi^2 \times 200, 000 \times 113 \times 10^6}{(1.0 \times 3, 600)^2} = 17, 211 \text{ kN}
\]

\[
U_{1x} = \frac{1.0}{1 - \frac{900}{17, 211}} = 1.06
\]

**Bending Capacity**

\[
\omega_2 = 1.0
\]

\[
EI_y GJ = 200, 000 \times 38.8 \times 76.92 \times 575 = 343.2 \times 10^{21} \text{ MPa}^2 \text{ mm}^8
\]

\[
\left( \frac{nE}{L} \right)^2 I_y C_w = \left( \frac{\pi \times 200, 000}{3, 600} \right)^2 38.8 \times 553 = 653.6 \times 10^{21} \text{ MPa}^2 \cdot \text{ mm}^8
\]

\[
M_u = \frac{\omega_2 n^2}{L} \sqrt{EI_y GJ + \left( \frac{nE}{L} \right)^2 I_y C_w} = \frac{1.0 \times \pi \times 3, 600}{3, 600} \sqrt{343.2 \times 10^{21} + 653.6 \times 10^{21}} = 871 \times 10^6 \text{ N mm} = 871 \text{ kN mm}
\]

Plastic moment capacity:

\[
M_p = Z_x F_y = 985 \times 10^3 \times 350 = 345 \times 10^6 \text{ N mm} = 345 \text{ kN m}
\]

\[
M_u = 871 \text{ kN mm} > 0.67 \times M_p = 0.67 \times 345 = 231 \text{ kN m}
\]

Therefore, the moment capacity is calculated as:

\[
M_r = 1.15 \phi M_p \left( 1 - \frac{0.28 M_p}{M_u} \right) = 1.15 \times 0.9 \times 345 \left( 1 - \frac{0.28 \times 345}{871} \right) = 317 \text{ kN m}
\]
Check the capacity based on the strength cross-section:

\[ M_{rx} = 0.9 \times 985 \times 10^3 \times 350 = 310 \text{ kN} \cdot \text{m} \]

This governs capacity.

**Combined Stress Ratio**

\[ \frac{900}{2.216} + \frac{0.85 \times 1.06 \times 180}{310} = 0.406 + 0.523 = 0.929 < 1.0 \]

**Comparison**

**Table 511: Verification Problem comparison**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>((KL/r)_x)</td>
<td>33</td>
<td>32.624</td>
<td>negligible</td>
</tr>
<tr>
<td>((KL/r)_y)</td>
<td>56</td>
<td>55.675</td>
<td>negligible</td>
</tr>
<tr>
<td>(C_r) (kN)</td>
<td>2,216</td>
<td>2,220</td>
<td>negligible</td>
</tr>
<tr>
<td>(C_e) (kN)</td>
<td>17,211</td>
<td>17,200</td>
<td>negligible</td>
</tr>
<tr>
<td>Stress Ratio</td>
<td>0.929</td>
<td>0.926</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**Note:** Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Shear Capacity Combined Stresses.STD is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.00e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
```
G 7.692e+07
END DEFINE MATERIAL
*MEMBER PROPERTY CANADIAN
*1 TABLE ST W250X73
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
2 ENFORCED BUT FY MX MZ
*2 ENFORCED BUT FX FZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
1 FY 900 MZ 180
2 FY -900 MZ -180
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
SSY 1 ALL
SSZ 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD.Pro Output

1

*****************************************************************************
*                             STAAD.Pro CONNECT Edition                   *
*                             Version 22.01.00.**                           *
*                             Proprietary Program of                      *
*                             Bentley Systems, Inc.                        *
*                             Date= APR 14, 2019                          *
*                             Time= 23: 7:38                             *
*                             Licensed to: Bentley Systems Inc             *
*****************************************************************************

1. STAAD SPACE
INPUT FILE: CSA S16-09 - Shear Capacity Combined Stresses.STD
  2. START JOB INFORMATION
  3. ENGINEER DATE 21-MAR-19
  4. END JOB INFORMATION
  5. INPUT WIDTH 79
  6. UNIT METER KN
  7. JOINT COORDINATES
  8. 1 0 0 0; 2 0 3.6 0
  9. MEMBER INCIDENCES
 10. 1 1 2
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 2.00E+08
14. POISSON 0.3
15. DENSITY 76.92
16. ALPHA 1.2E-05
17. DAMP 0.03
18. TYPE STEEL
19. STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
20. G 7.692E+07
21. END DEFINE MATERIAL
22. *MEMBER PROPERTY CANADIAN
23. *1 TABLE ST W250X73
24. MEMBER PROPERTY CANADIAN
25. *1 TABLE ST W250X73
26. CONSTANTS
27. MATERIAL STEEL ALL
28. SUPPORTS
29. *1 ENFORCED BUT MX MY MZ
30. 1 PINNED
31. 2 ENFORCED BUT FY MX MZ
32. *2 ENFORCED BUT FX FZ
33. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
34. JOINT LOAD
35. 1 FY 900 MZ 180
36. 2 FY -900 MZ -180
37. PERFORM ANALYSIS
   STAAD SPACE

2
PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       9
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

38. PRINT MEMBER PROPERTIES ALL
MEMBER PROPERTIES. UNIT - CM

3

4

SUPPORT REACTIONS -UNIT KN METE STRUCTURE TYPE = SPACE

************** END OF LATEST ANALYSIS RESULT **************
40. PARAMETER 1
41. CODE CANADIAN 2009
42. SSY 1 ALL
43. SSZ 1 ALL
44. FYLD 350000 ALL
45. FU 450000 ALL
46. TRACK 2 ALL
47. CHECK CODE ALL

STEEL DESIGN
STAD SPACE -- PAGE NO.

5

STAAD.PRO CODE CHECKING - S16-09 (v1.1)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
-----------------------------------------------
START OF DESIGN OUTPUT OF MEMBER 1

MEMBER NO: 1 CRITICAL RATIO: 0.926(PASS) LOAD: 1
LOCATION (MET): 0.00 CONDITION: Cl. 13.8.2
SECTION: ST W250X73 (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 0.926(PASS) LOAD CASE: 1 LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
DESIGN FORCES: Fx: 900.00(C) Fy: 0.00 Fz: 0.00
Mx: 0.00E+00 My: 0.00E+00 Mz: 1.80E+02
UNIT: CM
SECTION PROPERTIES: AZZ: 72.136 AYY: 20.537 CW: 552900.312
SZZ: 893.281 SYY: 305.512 IZZ: 11300.001 IYY: 3880.000
UNIT: NEW MM
MATERIAL PROPERTIES: FYLD: 350.000 FU: 450.000
ACTUAL MEMBER LENGTH(MET): 3.600
PARAMETERS: KZ: 1.000 KY: 1.000 NSF: 1.000 SLF: 1.000
SLENDERNESS: ACTUAL SLENDERNESS RATIO: 55.675 LOAD: 1 LOC.
(MET): 0.000 ALLOWABLE SLENDERNESS RATIO: 200.000

SECTION CLASS:
COMPRESSION: Class 1
FLEXURE: Class 2
FLANGE: Class 2
WEB: Class 1
UNIT: KN MET
TENSION:
LOCATION(MET):
YIELDING: 0.000 2923.200 0.000 Cl. 13.2
RUPTURE: 0.000 3132.000 0.000 Cl. 13.2

1 0.000
1 0.000

STAAD SPACE -- PAGE NO.

6

STAAD.PRO CODE CHECKING - S16-09 (v1.1)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
COMPRESSION:
FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
MAJOR: 900.000 2709.553 0.332 Cl. 13.3
<table>
<thead>
<tr>
<th></th>
<th>Minor:</th>
<th>Major:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>900.000</td>
</tr>
<tr>
<td>1</td>
<td>0.000</td>
<td>92.800</td>
</tr>
<tr>
<td>1</td>
<td>0.000</td>
<td>92.800</td>
</tr>
</tbody>
</table>

Shear:

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Force:</th>
<th>Capacity:</th>
<th>Ratio:</th>
<th>Criteria:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Location (Met):

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
<th>INTERMEDIATE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Ag: (CM)</td>
</tr>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Fe: (N MM)</td>
</tr>
<tr>
<td></td>
<td>0.000</td>
<td></td>
</tr>
</tbody>
</table>

Location (Met):

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
<th>INTERMEDIATE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Ag: (CM)</td>
</tr>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Fe: (N MM)</td>
</tr>
<tr>
<td></td>
<td>0.000</td>
<td></td>
</tr>
</tbody>
</table>

Yielding:

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
<th>INTERMEDIATE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Aw: (CM)</td>
</tr>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Kv:</td>
</tr>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Ka:</td>
</tr>
<tr>
<td></td>
<td>0.000</td>
<td></td>
</tr>
</tbody>
</table>

Location (Met):

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
<th>INTERMEDIATE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Aw: (CM)</td>
</tr>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Kv:</td>
</tr>
<tr>
<td>0.000</td>
<td>0.000</td>
<td>Ka:</td>
</tr>
<tr>
<td></td>
<td>0.000</td>
<td></td>
</tr>
</tbody>
</table>

Unit: KN MET

FLEXURE AND AXIAL TENSION:

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Location (Met):

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Member Strength:

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Flexure and Axial Compression:

<table>
<thead>
<tr>
<th>Minor:</th>
<th>Major:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
V.CSA S16-09 - Short Column Compression

Determine the factored compressive resistance of a short column.

Reference

Problem
A W250x73 section is used for a pedestal with a height of 1.1 m.
The material used is G40.21 350W steel ($F_y = 350$ MPa) (FYLD)

Calculations
The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.
Local Element Buckling

Evaluate the slenderness effects of the column web:

\[
\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8
\]

Evaluate the slenderness effects of the column flanges:

\[
\frac{b}{2r} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7
\]

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume \( K = 1.0, \)

\[
\frac{L}{r_x} = \frac{1.10}{110} = 10
\]

\[
\frac{L}{r_y} = \frac{1.10}{64.6} = 17
\]

The largest slenderness ratio controls, thus the slenderness factor for buckling about the minor axis:

\[
\lambda = \left(\frac{KL}{r}\right)_{\text{max}} \sqrt{\frac{F_y}{n^2E}} = \frac{17\sqrt{\frac{350}{n^2200,000}}}{n} = 0.227
\]

When \( n = 1.34 \) (Group 2 W shape),

\[
C_r = \phi AF_y (1 + \lambda^{2n})^{-1/n} = 0.90 \times 9, 280 \times 350 \left[1 + 0.227^{(2 \times 1.34)}\right]^{-1/1.34} = 2.883 \times (10)^6 \text{ N} = 2, 883 \text{ kN}
\]

Comparison

Table 512: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/r, major</td>
<td>10</td>
<td>9.968</td>
<td>negligible</td>
</tr>
<tr>
<td>KL/r, minor</td>
<td>17</td>
<td>17.012</td>
<td>negligible</td>
</tr>
<tr>
<td>( \lambda ), critical</td>
<td>0.227</td>
<td>0.224</td>
<td>1.3%</td>
</tr>
<tr>
<td>( C_r ) (kN)</td>
<td>2,883</td>
<td>2,884.358</td>
<td>negligible</td>
</tr>
</tbody>
</table>

STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Short Column Compression.STD is typically installed with the program.
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1.1 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
*1 TABLE ST L203x203x25
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
*2 FY -100
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD.Pro Output

1

******************************************************************************************
* *
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 23: 7:43 *
* *
* Licensed to: Bentley Systems Inc *
******************************************************************************************

1. STAAD SPACE
INPUT FILE: CSA S16-09 - Short Column Compression.STD
2. START JOB INFORMATION  
3. ENGINEER DATE 21-MAR-19  
4. END JOB INFORMATION  
5. INPUT WIDTH 79  
6. UNIT METER KN  
7. JOINT COORDINATES  
8. 1 0 0 0; 2 0 1.1 0  
9. MEMBER INCIDENCES  
10. 1 1 2  
11. DEFINE MATERIAL START  
12. ISOTROPIC STEEL  
13. E 2.05E+08  
14. POISSON 0.3  
15. DENSITY 76.92  
16. ALPHA 1.2E-05  
17. DAMP 0.03  
18. TYPE STEEL  
19. *STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2  
20. G 7.692E+07  
21. END DEFINE MATERIAL  
22. MEMBER PROPERTY CANADIAN  
23. *1 TABLE ST L203X203X25  
24. 1 TABLE ST W250X73  
25. CONSTANTS  
26. MATERIAL STEEL ALL  
27. SUPPORTS  
28. 1 FIXED  
29. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1  
30. JOINT LOAD  
31. 2 FY -0.001  
32. *2 FY -100  
33. PERFORM ANALYSIS  

--- PAGE NO. 2 ---

PROBLEM STATISTICS

-----------------------------------
NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1  
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0  
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      1  
Using 64-bit analysis engine.  
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER  
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       6  
TOTAL LOAD COMBINATION CASES =     0 SO FAR.  

--- PAGE NO. 3 ---

MEMBER PROPERTIES. UNIT - CM

-----------------  
MEMB PROFILE AX/ IZ/ IV/ IX/  
AY AZ SZ SY  
1 ST W250X73 92.80 11300.00 3880.00 57.50  
21.76 48.09 893.28 305.51  
*********** END OF DATA FROM INTERNAL STORAGE ***********  

--- PAGE NO. 4 ---

34. PRINT MEMBER PROPERTIES ALL  
35. PRINT SUPPORT REACTION ALL  

--- PAGE NO. 5 ---

36. END
SUPPORT REACTIONS -UNIT KN METE STRUCTURE TYPE = SPACE

-----------------

JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM Z
1 1 0.00 0.00 0.00 0.00 0.00 0.00

*************** END OF LATEST ANALYSIS RESULT ***************

36. PARAMETER 1
37. CODE CANADIAN 2009
38. FYLD 350000 ALL
39. FU 450000 ALL
40. TRACK 2 ALL
41. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE

STAAD.PRO CODE CHECKING - S16-09 (v1.1)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)

-------------------- START OF DESIGN OUTPUT OF MEMBER 1

MEMBER NO: 1 CRITICAL RATIO: 0.000(PASS) LOAD: 1
LOCATION (MET): 0.00 CONDITION: Cl. 13.8.2
SECTION: ST W250X73 (CANADIAN SECTIONS)
UNIT: KN MET

STRENGTH CHECKS:
CRITICAL RATIO: 0.000(PASS) LOAD CASE: 1 LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
DESIGN FORCES: Fx: 0.00(C) Fy: 0.00 Fz: 0.00
Mx: 0.00E+00 My: 0.00E+00 Mz: 0.00E+00
UNIT: CM

SECTION PROPERTIES: AZZ: 72.136 AYY: 20.537 CW: 552900.312
SZZ: 893.281 SYY: 305.512 IZZ: 11300.001 IYY: 3880.000
UNIT: NEW MM

MATERIAL PROPERTIES: FYLD: 350.000 FU: 450.000
ACTUAL MEMBER LENGTH(MET): 1.100
PARAMETERS: KZ: 1.000 KY: 1.000 NSF: 1.000 SLF: 1.000

SLENDERNESS: ACTUAL SLENDERNESS RATIO: 17.012 LOAD: 1 LOC.
(MET): 0.000 ALLOWABLE SLENDERNESS RATIO: 200.000

SECTION CLASS:
COMPRESSION: Class 1
FLEXURE: Class 2
FLANGE: Class 2
WEB: Class 1

UNIT: KN MET

TENSION:
LOCATION(MET): FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
1 YIELDING: 0.000 2923.200 0.000 Cl. 13.2
1 RUPTURE: 0.000 3132.000 0.000 Cl. 13.2

STAAD SPACE

STAAD.PRO CODE CHECKING - S16-09 (v1.1)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
<table>
<thead>
<tr>
<th>COMPRESSION:</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
<th>LOAD CASE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION(MET):</td>
<td>MAJOR:</td>
<td>0.001</td>
<td>2913.816</td>
<td>0.000</td>
<td>Cl. 13.3</td>
</tr>
<tr>
<td>MINOR:</td>
<td>0.001</td>
<td>2884.358</td>
<td>0.000</td>
<td>Cl. 13.3</td>
<td></td>
</tr>
<tr>
<td>INTERMEDIATE:</td>
<td>Ag:(CM):</td>
<td>FL/r:</td>
<td>Fe:(N MM)</td>
<td>1</td>
<td>n:</td>
</tr>
<tr>
<td>MAJOR:</td>
<td>92.800</td>
<td>9.968</td>
<td>20360.980</td>
<td>0.131</td>
<td>1.340</td>
</tr>
<tr>
<td>MINOR:</td>
<td>92.800</td>
<td>17.012</td>
<td>6991.204</td>
<td>0.224</td>
<td>1.340</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SHEAR:</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
<th>LOAD CASE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION(MET):</td>
<td>MAJOR:</td>
<td>0.000</td>
<td>452.349</td>
<td>0.000</td>
<td>Cl. 13.4.1.1</td>
</tr>
<tr>
<td>MINOR:</td>
<td>0.000</td>
<td>1363.370</td>
<td>0.000</td>
<td>AISC G2-1</td>
<td></td>
</tr>
</tbody>
</table>

| MAJOR: | 20.537 | 5.552 | 0.224 | 489.474 |
| MINOR: | 72.136 | |

<table>
<thead>
<tr>
<th>YIELDING:</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
<th>LOAD CASE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION(MET):</td>
<td>MAJOR:</td>
<td>0.00E+00</td>
<td>3.10E+02</td>
<td>0.000</td>
<td>Cl. 13.5(a)</td>
</tr>
<tr>
<td>MINOR:</td>
<td>0.00E+00</td>
<td>1.46E+02</td>
<td>0.000</td>
<td>Cl. 13.5(a)</td>
<td></td>
</tr>
<tr>
<td>INTERMEDIATE:</td>
<td>Mp:</td>
<td>Se:</td>
<td>My:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MAJOR:</td>
<td>3.45E+02</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MINOR:</td>
<td>1.62E+02</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LTB BUCK:</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
<th>LOAD CASE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION(MET):</td>
<td>MAJOR:</td>
<td>0.00E+00</td>
<td>3.55E+02</td>
<td>0.000</td>
<td>Cl. 13.6(a)(i)</td>
</tr>
<tr>
<td>INTERMEDIATE:</td>
<td>Iyc(CM):</td>
<td>w 2:</td>
<td>w 3:</td>
<td>b x:</td>
<td>Mu:</td>
</tr>
<tr>
<td>MAJOR:</td>
<td>0.00E+00</td>
<td>2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>1.98E+04</td>
</tr>
<tr>
<td>UNIT: KN MET</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>INTERACTION:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
<th>LOAD CASE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION(MET):</td>
<td>FLEXURE AND AXIAL TENSION:</td>
<td>C/S STRENGTH:</td>
<td>0.000</td>
</tr>
<tr>
<td>FLEXURE AND AXIAL COMPRESSION:</td>
<td>C/S STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.2</td>
</tr>
<tr>
<td>MEMBER STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.2</td>
<td>1</td>
</tr>
<tr>
<td>LT8 STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.2</td>
<td>1</td>
</tr>
<tr>
<td>FLEX AND SHEAR:</td>
<td>0.000</td>
<td>Cl. 14.6(a)</td>
<td>1</td>
</tr>
<tr>
<td>BIAXIAL FLEX:</td>
<td>0.000</td>
<td>Cl. 13.8</td>
<td>1</td>
</tr>
</tbody>
</table>

| 3.10E+02 |

STAAD SPACE
V.CSA S16-09 - Slender Column Compression

Determine the factored compressive resistance of a slender column.

Reference

Problem
A W250x73 section is used for a pedestal with a height of 11.0 m.
The material used is G40.21 350W steel ($F_y = 350$ MPa) (FYLD)

Calculations
The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.
Local Element Buckling

Evaluate the slenderness effects of the column web:

\[ \frac{h}{w} = \frac{225}{8.8} = 26.2 < \frac{670}{\sqrt{350}} = 35.8 \]

Evaluate the slenderness effects of the column flanges:

\[ \frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7 \]

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume \( K = 1.0 \),

\[ \frac{L}{r_x} = \frac{11,000}{110} = 100 \]

\[ \frac{L}{r_y} = \frac{11,000}{64.6} = 170 \]

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

\[ F_e = \frac{\pi^2 \times 200,000}{(1.0 \times 170)^2} = 68.3 \text{ kN} \]

\[ \lambda = \sqrt{\frac{F_y}{F_e}} = \sqrt{\frac{350}{68.3}} = 2.26 \]

When \( n = 1.34 \) (Group 2 W shape),

\[ C_r = \phi A F_y \left(1 + \lambda^2 n\right)^{-1/n} = 0.90 \times 9, 280 \times 350 \left[1 + 2.26^{(2 \times 1.34)}\right]^{-1/1.34} = 0.529 \times (10)^6 \text{ N} = 529 \text{ kN} \]

Comparison

Table 513: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/r, major</td>
<td>100</td>
<td>99.684</td>
<td>negligible</td>
</tr>
<tr>
<td>KL/r, minor</td>
<td>170</td>
<td>170.118</td>
<td>negligible</td>
</tr>
<tr>
<td>( \lambda ), critical</td>
<td>2.26</td>
<td>2.237</td>
<td>1.0%</td>
</tr>
<tr>
<td>( C_r ) (kN)</td>
<td>529</td>
<td>538.161</td>
<td>1.7%</td>
</tr>
</tbody>
</table>
STAAD Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Slender Column Compression.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 11 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Pro Output

<table>
<thead>
<tr>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
</tr>
</tbody>
</table>

************************************************
*                                              *
*                     STAAD.Pro CONNECT Edition  *
*  Version 22.01.00.**                           *
*  Proprietary Program of                       *
*  Bentley Systems, Inc.                        *
*                                              *
************************************************
1. STAAD SPACE
INPUT FILE: CSA S16-09 - Slender Column Compression.STD
2. START JOB INFORMATION
3. ENGINEER DATE 22-MAR-19
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER KN
7. JOINT COORDINATES
8. 1 0 0 0; 2 0 11 0
9. MEMBER INCIDENCES
10. 1 1 2
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 2.05E+08
14. POISSON 0.3
15. DENSITY 76.92
16. ALPHA 1.2E-05
17. DAMP 0.03
18. TYPE STEEL
19. *STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
20. G 7.692E+07
21. END DEFINE MATERIAL
22. MEMBER PROPERTY CANADIAN
23. 1 TABLE ST W250X73
24. CONSTANTS
25. MATERIAL STEEL ALL
26. SUPPORTS
27. 1 FIXED
28. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
29. JOINT LOAD
30. 2 FY -0.001
31. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      1
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       6
TOTAL LOAD COMBINATION CASES =     0 SO FAR.
32. PRINT MEMBER PROPERTIES ALL
33. MEMBER PROPERTIES. UNIT - CM

-- PAGE NO.

STAAD SPACE

-- PAGE NO.

STAAD SPACE

-- PAGE NO.
33. PRINT SUPPORT REACTION ALL
SUPPORT REACTION ALL
STAAD SPACE

4
SUPPORT REACTIONS -UNIT KN METE STRUCTURE TYPE = SPACE
-----------------
JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM Z
1 1 0.00 0.00 0.00 0.00 0.00 0.00

34. PARAMETER 1
35. CODE CANADIAN 2009
36. FYLD 350000 ALL
37. FU 450000 ALL
38. TRACK 2 ALL
39. CHECK CODE ALL
STEEL DESIGN
STAAD SPACE

5
STAAD.PRO CODE CHECKING - S16-09 (v1.1)
*******************************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
----------------------- START OF DESIGN OUTPUT OF MEMBER 1
-----------------------
MEMBER NO: 1 CRITICAL RATIO: 0.000(PASS) LOAD: 1
LOCATION (MET): 0.00 CONDITION: Cl. 13.8.2
SECTION: ST W250X73 (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 0.000(PASS) LOAD CASE: 1 LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
DESIGN FORCES: Fx: 0.00(C) Fy: 0.00 Fz: 0.00
Mx: 0.00E+00 My: 0.00E+00 Mz: 0.00E+00
UNIT: CM
SECTION PROPERTIES: AZZ: 72.136 AYY: 20.537 CW: 552900.312
SZZ: 893.281 SYY: 305.512 IZZ: 11300.001 IYY: 3880.000
UNIT: NEW MM
MATERIAL PROPERTIES: FYLD: 350.000 FU: 450.000
ACTUAL MEMBER LENGTH(MET): 11.000
PARAMETERS: KZ: 1.000 KY: 1.000 NSF: 1.000 SLF: 1.000
SLENDERNESS: ACTUAL SLENDERNESS RATIO: 170.118 LOAD: 1 LOC.
(MET): 0.000 ALLOWABLE SLENDERNESS RATIO: 200.000
SECTION CLASS:
COMPRESSION: Class 1
FLEXURE: Class 2
FLANGE: Class 2
WEB: Class 1
UNIT: KN MET
TENSION:
LOCATION(MET):
YIELDING: 0.000 2923.200 0.000 Cl. 13.2
RUPTURE: 0.000 3132.000 0.000 Cl. 13.2
### All Units Are KN MET (Unless Otherwise Noted)

<table>
<thead>
<tr>
<th>Location (MET)</th>
<th>Major</th>
<th>Minor</th>
<th>Inter</th>
<th>Shear</th>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.001</td>
<td>0.001</td>
<td>0.000</td>
<td>0.000</td>
<td>1266.716</td>
<td>538.161</td>
<td>0.000</td>
<td>Cl. 13.3</td>
<td>1.340</td>
</tr>
<tr>
<td>Minor</td>
<td>0.000</td>
<td>0.000</td>
<td>1.340</td>
<td>1.340</td>
<td>538.161</td>
<td>538.161</td>
<td>0.000</td>
<td>Cl. 13.3</td>
<td>1.340</td>
</tr>
<tr>
<td>Intermediate</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>92.800</td>
<td>99.684</td>
<td>203.610</td>
<td>1.311</td>
<td>1.340</td>
</tr>
<tr>
<td>Major</td>
<td>0.000</td>
<td>0.000</td>
<td>1.340</td>
<td>1.340</td>
<td>99.684</td>
<td>99.684</td>
<td>0.000</td>
<td>Cl. 13.4</td>
<td>1.340</td>
</tr>
<tr>
<td>Minor</td>
<td>0.000</td>
<td>0.000</td>
<td>1.340</td>
<td>1.340</td>
<td>99.684</td>
<td>99.684</td>
<td>0.000</td>
<td>Cl. 13.4</td>
<td>1.340</td>
</tr>
</tbody>
</table>

---

### Verification Examples

**V.09 Steel Design**

**STAAD.Pro**

**User Manual**
V.CSA S16-09 - Wide Flange Capacity Combined Stresses

Check the capacity of a wide-flange column under combined stresses.

Reference


Problem

The W250x73 column has the following concentrated loads applied at the top of the column:

\[ C_f = 900 \text{ kN} \]

\[ M_{fx} = 180 \text{ kN m} \]
The material used is G40.21 350W steel ($F_y = 350 \text{ MPa}$) (FYLD). The column is assumed to be braced laterally against lateral-torsional buckling (LAT 1).

$L = 3.6 \text{ m}$

Calculations

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Axial Capacity

Determine the buckling load of the column, assuming that weak axis bucking will not control due to bracing:

$$\frac{(KL/r)_x}{1.0(600)} = \frac{1.0(600)}{116} = 32.7$$

The largest slenderness ratio controls:

$$F_{ex} = \frac{n^2 E}{(KL/r)_x} = \frac{n^2 (200,000)}{(32.7)^2} = 1,847 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_{ex}}{F_{ey}}} = \sqrt{\frac{350}{1,847}} = 0.435$$

$$C_r = \phi A F_y (1 + \lambda^{2n})^{-1/n} = 0.90 \times 9, 280 \times 350 [1 + 0.435^{(2 \times 1.34)}]^{-1/1.34} = 2,716 \times (10)^6 \text{ N} = 2,716 \text{ kN}$$

Moment Modifier

$$\kappa = -0/180 = 0$$

$$\omega_1 = 0.6 - 0.4 \kappa = 0.6 \leq 1.0$$

Calculate the elastic buckling load for the bending axis:

$$C_e = \frac{n^2 	imes 200,000 \times 113 \times 10^6}{(1.0 \times 3,600)^2} = 17,211 \text{ kN}$$

$$U_{1x} = \frac{0.6}{1 - \left(\frac{900}{17,210}\right)} = 0.633$$

Bending Capacity

From a previous example, the W250x73 is a Class 2 section. It is laterally supported along its length, thus:
\[ M_{1x} = 0.9 \times 985 \times 10^3 \times 350 = 310 \text{ kN m} \]

**Combined Stress Ratio**

\[ \frac{900}{2} \times \frac{716}{310} + \frac{0.85 \times 0.633 \times 180}{310} = 0.331 + 0.312 = 0.644 < 1.0 \]

This is the check for overall member strength. However, there is another case that actually controls in this instance: checking for cross sectional strength.

\[ C_r = 0.9 \times 9,280 \times 350 = 2,923 \text{ kN} \]

\[ U_{1x} = 1.0 \]

\[ \frac{900}{2} \times \frac{923}{310} + \frac{0.85 \times 1.0 \times 180}{310} = 0.308 + 0.494 = 0.801 < 1.0 \]

**Comparison**

**Table 514: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>((K L / r)_{x})</td>
<td>33</td>
<td>32.624</td>
<td>negligible</td>
</tr>
<tr>
<td>((K L / r)_{y})</td>
<td>56</td>
<td>55.675</td>
<td>negligible</td>
</tr>
<tr>
<td>(C_r) (kN)</td>
<td>2,716</td>
<td>2,715.935</td>
<td>negligible</td>
</tr>
<tr>
<td>(C_e) (kN)</td>
<td>17,211</td>
<td>17,600</td>
<td>2.3%</td>
</tr>
<tr>
<td>Stress Ratio</td>
<td>0.644</td>
<td>0.643</td>
<td>none</td>
</tr>
<tr>
<td>Stress Ratio (Critical)</td>
<td>0.801</td>
<td>0.801</td>
<td>none</td>
</tr>
</tbody>
</table>

**Note:** Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Wide Flange Capacity Combined Stresses.STD is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES 1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES 1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
```
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
2 ENFORCED BUT FY MX MZ
*2 ENFORCED BUT FX FZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
1 FY 900
*1 MZ 180
2 FY -900 MZ 180
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
LAT 1 ALL
SSZ 1 ALL
SSY 1 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

**STAAD.Pro Output**

<table>
<thead>
<tr>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
</tr>
</tbody>
</table>

--------------------------------------------------------------------------------------
*                        *
* STAAD.Pro CONNECT Edition                        *
* Version 22.01.00.**                                *
* Proprietary Program of                            *
* Bentley Systems, Inc.                              *
* Date= APR 14, 2019                                 *
* Time= 23: 7:53                                     *
* Licensed to: Bentley Systems Inc                  *
--------------------------------------------------------------------------------------
1. STAAD SPACE
INPUT FILE: CSA S16-09 - Wide Flange Capacity Combined Stresses.STD
2. START JOB INFORMATION
3. ENGINEER DATE 17-MAR-19
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER KN
7. JOINT COORDINATES
   1 0 0 0; 2 0 3.6 0
8. MEMBER INCIDENCES
   1 1 2
9. DEFINE MATERIAL START
10. ISOTROPIC STEEL
11. E 2.05E+08
12. POISSON 0.3
13. DENSITY 76.92
14. ALPHA 1.2E-05
15. DAMP 0.03
16. TYPE STEEL
17. STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
18. G 7.692E+07
19. END DEFINE MATERIAL
20. MEMBER PROPERTY CANADIAN
21. 1 TABLE ST W250X73
22. CONSTANTS
23. MATERIAL STEEL ALL
24. SUPPORTS
25. *1 ENFORCED BUT MX MY MZ
26. 1 PINNED
27. 2 ENFORCED BUT FY MX MZ
28. *2 ENFORCED BUT FX FZ
29. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
30. JOINT LOAD
31. 1 FY 900
32. 2 FY -900 MZ 180
33. PERFORM ANALYSIS
34. STAAD SPACE
35. -- PAGE NO.
36. PROBLEM STATISTICS
37. NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
38. NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
39. NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
40. Using 64-bit analysis engine.
41. SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
42. TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       9
43. TOTAL LOAD COMBINATION CASES =     0  SO FAR.
44. PRINT MEMBER PROPERTIES ALL
45. STAAD SPACE
46. -- PAGE NO.
47. MEMBER PROPERTIES. UNIT - CM
48. -----------------------------
49. MEMB  PROFILE              AX/          IZ/          IY/        IX/
50.      AY           AZ           SZ         SY
51. 1  ST W250X73           92.80     11300.00      3880.00       57.50
52. 21.76        48.09       893.28      305.51
53. ************ END OF DATA FROM INTERNAL STORAGE ************
54. PRINT SUPPORT REACTION ALL
55. SUPPORT REACTION ALL
56. STAAD SPACE
57. -- PAGE NO.
### SUPPORT REACTIONS -UNIT KN METE STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-50.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>50.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

39. PARAMETER 1
40. CODE CANADIAN 2009
41. FYLD 350000 ALL
42. LAT 1 ALL
43. SSZ 1 ALL
44. SSY 1 ALL
45. FU 450000 ALL
46. TRACK 2 ALL
47. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE

--- PAGE NO. 5

STAAD.PRO CODE CHECKING - S16-09 (v1.1)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)

------------------ START OF DESIGN OUTPUT OF MEMBER 1 ------------------

MEMBER NO: 1 CRITICAL RATIO: 0.801(PASS) LOAD: 1 LOCATION (MET): 3.60 CONDITION: Cl. 13.8.2

SECTION: ST W250X73 (CANADIAN SECTIONS)

UNIT: KN MET

STRENGTH CHECKS:
CRITICAL RATIO: 0.801(PASS) LOAD CASE: 1 LOCATION (MET): 3.60 CONDITION: Cl. 13.8.2

DESIGN FORCES: Fx: 900.00(C) Fy: 50.00 Fz: 0.00
Mx: 0.00E+00 My: 0.00E+00 Mz: -1.80E+02

UNIT: CM

SECTION PROPERTIES: AZZ: 72.136 AYY: 20.537 CW:
552900.312
SZZ: 893.281 SYY: 305.512 IZZ: 11300.001 IYY: 3880.000

UNIT: NEW MM

MATERIAL PROPERTIES: FYLD: 350.000 FU: 450.000

ACTUAL MEMBER LENGTH(MET): 3.600

PARAMETERS: KZ: 1.000 KY: 1.000 NSF: 1.000 SLF: 1.000

SLENDERNESS: ACTUAL SLENDERNESS RATIO: 55.675 LOAD: 1 LOC.
(MET): 0.000 ALLOWABLE SLENDERNESS RATIO: 200.000

SECTION CLASS:
COMPRESS: Class 1
FLEXURE: Class 2
FLANGE: Class 2
WEB: Class 1

UNIT: KN MET

TENSION:
FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
YIELDING: 0.000 2923.200 0.000 Cl. 13.2
RUPTURE: 0.000 3132.000 0.000 Cl. 13.2

--- PAGE NO.
### Verification Examples
#### V.09 Steel Design

**STAAD.PRO CODE CHECKING - (S16-09) v1.1**

- **ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)**
- **LOCATION(MET):**
  - **MAJOR:** 900.000 2715.935 0.331 Cl. 13.3
  - **MINOR:** 900.000 2233.940 0.403 Cl. 13.3
  - **INTERMEDIATE:**
    - **MAJOR:** Ag: (CM) 92.800 32.624 1900.987 0.429 1.340
    - **MINOR:** 92.800 55.675 652.728 0.732 1.340
- **SHEAR:**
  - **MAJOR:** 50.000 452.349 0.111 Cl. 13.4.1.1
  - **MINOR:** 0.000 1363.370 0.000 AISC G2-1
  - **INTERMEDIATE:**
    - **MAJOR:** 20.537 5.360 0.070 480.943
    - **MINOR:**
- **UNIT:** KN MET
- **YIELDING:**
  - **MAJOR:** 1.80E+02 3.10E+02 0.580 Cl. 13.5(a)
  - **MINOR:** 0.00E+00 1.46E+02 0.000 Cl. 13.5(a)
  - **INTERMEDIATE:**
    - **MAJOR:**
    - **MINOR:**
- **INTERACTION:**
  - **FLEXURE AND AXIAL TENSION:**
    - **C/S STRENGTH:** 0.580 Cl. 13.9.1 1 3.600
  - **FLEXURE AND AXIAL COMPRESSION:**
    - **C/S STRENGTH:** 0.801 Cl. 13.8.2 1 3.600
    - **MEMBER STRENGTH:** 0.715 Cl. 13.8.2 1 0.000
  - **FLEX AND SHEAR:**
    - **INTERMEDIATE:**
      - **FLEXURE AND AXIAL TENSION:**
        - **C/S STRENGTH:**
        - **MEM STR:**
        - **FLEX SHEAR:**
          - **INTERMEDIATE:**
            - **FLEXURE AND AXIAL TENSION:**
              - **C/S STR:**
              - **MEM STR:**
              - **FLEX SHEAR:**

---

**STAAD SPACE**  -- PAGE NO.
V. CAN/CSA S16-14

The following are several verification examples for reference purposes. Since the S16-14 code is similar in many respects to the previous edition of the code (CAN/CSA S16-09), the solved examples of the 1994 edition of the CISC Handbook have been used as reference material for these examples.

V. CSA S16-14 - Axial Tension

Design of a double-angle member subject to tension load.

Reference

Problem
Design a tension diagonal in an all-welded truss (SNU 0 for welded). Attached to WT 265x61.5 chords (t_w = 13.1 mm).

\[ L = 4 \, \text{m} \]
\[ T_f = 630 \, \text{kN} \]

The material used is G40.21 300W steel \( (F_y = 300 \, \text{MPa}, F_u = 450 \, \text{MPa}) \) \( (\text{FYLD and FU}) \)

Calculations
Gross area is used for welded connections.

\[ T_r = \max\left\{ \phi_A F_y = 0.9 \times 300 \times A = 270\,A \right\}
\[ \phi_u A F_u = 0.75 \times 450 \times A = 338\,A \]
\[ T_r \geq T_f, \text{ so solve for } A: \]
\[ A = \frac{630 \times (10)^3}{270} = 2,333 \text{ mm}^2 \]

Select a pair of angles with at least this gross area: try a 2-76x64x9.5, with long legs back-to-back:
\[ A = 2,480 \text{ mm}^2 \]
\[ r_{\min} = 13.3 \text{ mm} \]

Calculate the effective net area due to shear lag. The average weld length is taken as \((120 \text{ mm} + 250 \text{ mm})/2 = 185 \text{ mm}\).

The long leg does not have a reduction in area shear lag, thus:
\[ A_{n2} = 1.00 \times (76 - 9.5) \times 9.5 = 632 \text{ mm}^2 \]

The outstanding (short) leg has an eccentricity of half the leg length, or \(64/2 = 32 \text{ mm}\).
\[ A_{n3} = \left(1 - \frac{32}{250}\right) \times 64 \times 9.5 = 530 \text{ mm}^2 \]

The total net area for the pair of angles for the rupture limit state is:
\[ A_{ne} = 632 + 530 = 1,162 \text{ mm}^2 \]

For the pair of angles, \(A_{ne} = 2 \times 1,1162 = 2,324 \text{ mm}^2\); the ratio of net section area to gross area = \(2,324 / 2,480 = 0.937 \text{ (NSF)}\).

Checking the yielding limit state:
\[ T_r = 0.90 \times 2,480 \times 300 = 670 \text{ kN} \]

Checking the rupture limit state:
\[ T_r = 0.75 \times 2,324 \times 450 = 784 \text{ kN} \]

Yielding governs, and the critical ratio is: \(630/670 = 0.94\).

### Comparison

Table 515: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(T_r ) (kN)</td>
<td>670</td>
<td>669.6</td>
<td>negligible</td>
</tr>
<tr>
<td>Critical Ratio</td>
<td>0.94</td>
<td>0.941</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**Note:** If the member SELECT facility is used in place of a code CHECK, STAAD.Pro will actually select a 2-L 102x89x6.4 member, which has a critical ratio of 0.997 (per Cl. 13.2) and an area of 2,340 mm\(^2\), thus more economical (the NSF used for this size is 0.923).
**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2014\CSA S16-14 - Axial Tension.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Apr-14
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE LD L76X64X9.5
*1 TABLE T W530x123
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FX 630
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
FYLD 300000 ALL
FU 450000 ALL
SNUG 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

**STAAD Output**

STAAD.PRO CODE CHECKING - S16-14 (v1.0)
******************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
------------------------ START OF DESIGN OUTPUT OF MEMBER 1

**Verification Examples**

V.09 Steel Design
MEMBER NO:     1    CRITICAL RATIO:  0.941(PASS)     LOAD:     1
LOCATION (MET): 0.00  CONDITION: Cl. 13.9.1
SECTION: LD    L76X64X9.5              (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO:  0.941(PASS)     LOAD CASE:     1 LOCATION (MET): 0.00
CONDITION: Cl. 13.9.1
DESIGN FORCES:   Fx:    630.00(T)    Fy:     0.00    Fz:      0.00
Mx:   0.00E+00      My:  0.00E+00     Mz:   0.00E+00
UNIT: CM
SECTION PROPERTIES:       AZZ:      11.195 AYY:      13.616 CW: 0.000
SZZ:      26.578 SYY:      26.229
IZZ:     137.969 IYY:     166.556
UNIT: NEW MM
MATERIAL PROPERTIES:      FYLD:      300.000   FU:      450.000
ACTUAL MEMBER LENGTH(MET):     4.000
PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF: 1.000
1.000
SLENDERNESS: ACTUAL SLENDERNESS RATIO:  169.796 LOAD:     1 LOC. (MET): 0.000
ALLOWABLE SLENDERNESS RATIO:  300.000
SECTION CLASS:
COMPRESSION:     Class 1
FLEXURE:         Class 3
FLANGE:          Class 3
UNIT: KN MET
TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
MAJOR:           0.000     141.791      0.000    Cl. 13.3
MINOR:           0.000     166.729      0.000    Cl. 13.3
INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)         l           n:
MAJOR:          24.800    169.796       70.178      2.068        1.340
MINOR:          24.800    154.539       84.719      1.882        1.340
FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
0.000     164.708      0.000    Cl. 13.3           1
INTERMEDIATE:   Fe:(N MM)  l             n:
83.506    1.895        1.340
SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
MAJOR:           0.000     226.444      0.000    Cl. 13.4.3
MINOR:           0.000     196.070      0.000    AISC G2-1
### Verification Examples

**V.09 Steel Design**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Fs:(N MM)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>MAJOR:</td>
<td>13.616</td>
<td>1.200</td>
<td>0.000</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>MINOR:</td>
<td>0.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**UNIT: KN MET**

#### Yielding

<table>
<thead>
<tr>
<th>LOCATION(MET):</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
<th>LOAD CASE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAJOR:</td>
<td>0.00E+00</td>
<td>7.18E+00</td>
<td>0.000</td>
<td>Cl. 13.5(b)</td>
<td></td>
</tr>
<tr>
<td>MINOR:</td>
<td>0.00E+00</td>
<td>7.08E+00</td>
<td>0.000</td>
<td>Cl. 13.5(b)</td>
<td></td>
</tr>
</tbody>
</table>

#### Lat Tor Buck

<table>
<thead>
<tr>
<th>LOCATION(MET):</th>
<th>FORCE:</th>
<th>CAPACITY:</th>
<th>RATIO:</th>
<th>CRITERIA:</th>
<th>LOAD CASE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAJOR:</td>
<td>0.00E+00</td>
<td>1.17E+01</td>
<td>0.000</td>
<td>Cl. 13.6(e)(i)</td>
<td></td>
</tr>
</tbody>
</table>

#### Staat Space

--- END OF DESIGN OUTPUT OF MEMBER 1 ---

--- Page No. ---
V. CSA S16-14 - Beam Bending

Determine the uniformly factored load that the member can resist.

Reference

Problem
A W310x52 beam spans 7.3 m. Both ends of the beam are supported by columns connected standard web angle connections.

The material used is G40.21 350W steel ($F_y = 350 \text{ MPa}$) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Section Classification

Evaluate the slenderness effects of the beam flanges:

$$b = \frac{167}{2 \times 13.2} = 6.3 < \frac{170}{\sqrt{350}} = 9.1$$

Evaluate the slenderness effects of the beam web:

$$h = \frac{318 - 2(13.2)}{7.6} = 38.4 < \frac{1,700}{\sqrt{350}} = 90.9$$

Both the beam flanges and web are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 5.7.

Bending Capacity

Factored moment resistance is:

$$M_r = \phi Z_x F_y = 0.90 \times 841 \times (10)^3 \times 350 = 265 \times (10)^3 \text{ N} \cdot \text{mm} = 265 \text{ kN} \cdot \text{m}$$

Equate this to the bending moment due to a uniformly distributed load and solve for the load:

$$M_r \geq \frac{w_f L^2}{8}$$

$$w_f = \frac{8M_r}{L^2} = \frac{8 \times 265}{(7.3)^2} = 39.8 \text{ kN/m}$$
Comparison

Table 516: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_r$ (kN·m)</td>
<td>265</td>
<td>265</td>
<td>none</td>
</tr>
<tr>
<td>$w_f$ (kN/m)</td>
<td>39.8</td>
<td>39.76*</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**Note:** (*) STAAD.Pro does calculate the allowable uniform load on the beam. Instead, this is done by applying a uniform load incrementally until the critical ratio is $\approx 1.0$. The, this uniform load is divided by the resulting critical ratio to normalize the distributed load capacity of the beam. Thus, $\frac{39.8}{1.001} = 39.67$ kN/m

**STAAD Input**

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X52
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -39.8
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
```
**STAAD Output**

**STAAD.PRO CODE CHECKING - S16-14 (v1.0)**

----------------------------------
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
----------------------------------

*MEMBER NO: 1 CRITICAL RATIO: 1.001(FAIL) LOAD: 1
LOCATION (MET): 3.65 CONDITION: Cl. 13.8
SECTION: ST W310X52 (CANADIAN SECTIONS)
UNIT: KN MET

STRENGTH CHECKS:
CRITICAL RATIO: 1.001(FAIL) LOAD CASE: 1 LOCATION (MET): 3.65
CONDITION: Cl. 13.8
DESIGN FORCES: Fx: 0.00(T) Fy: 0.00 Fz: 0.00
Mx: 0.00E+00 My: 0.00E+00 Mz: -2.65E+02
UNIT: CM

SECTION PROPERTIES: A ZZ: 44.088 A YY: 23.165 CW: 237980.875
S ZZ: 748.428 S YY: 123.353
I ZZ: 11900.002 I YY: 1030.000
UNIT: NEW MM

MATERIAL PROPERTIES: FYLD: 350.000 FU: 450.000
ACTUAL MEMBER LENGTH(MET): 7.300
PARAMETERS: K Z: 1.000 K Y: 1.000 NSF: 1.000 SLF: 1.000

SLENDERNESS: ACTUAL SLENDERNESS RATIO: 185.766 LOAD: 1 LOC.
(MET): 0.000
ALLOWABLE SLENDERNESS RATIO: 300.000

SECTION CLASS:
COMPRESSION: Class 4
FLEXURE MAJOR MINOR
SECTION: Class 1 Class 1
FLANGE: Class 1 Class 1
WEB: Class 1
UNIT: KN MET

TENSION: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
YIELDING: 0.000 2101.050 0.000 Cl. 13.2
1 0.000 RUPTURE: 0.000 2251.125 0.000 Cl. 13.2
1 0.000

-- PAGE NO.

STAAD SPACE

---
<table>
<thead>
<tr>
<th></th>
<th></th>
<th>Cl. 13.3.5(a)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>MINOR:</td>
<td>0.000</td>
<td>322.454</td>
<td>0.000</td>
</tr>
<tr>
<td>1</td>
<td>0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>INTERMEDIATE:</td>
<td>Ag:(CM)</td>
<td>KL/r:</td>
<td>Fe:(N MM)</td>
</tr>
<tr>
<td>MAJOR:</td>
<td>65.224</td>
<td>54.653</td>
<td>677.375</td>
</tr>
<tr>
<td>MINOR:</td>
<td>65.224</td>
<td>185.766</td>
<td>58.630</td>
</tr>
<tr>
<td>FLEX TOR BUCK:</td>
<td>FORCE:</td>
<td>CAPACITY:</td>
<td>RATIO:</td>
</tr>
<tr>
<td>LOCATION(MET):</td>
<td>0.000</td>
<td>1023.539</td>
<td>0.000</td>
</tr>
<tr>
<td>INTERMEDIATE:</td>
<td>Fe:(N MM)</td>
<td>l</td>
<td>n:</td>
</tr>
<tr>
<td>MAJOR:</td>
<td>-145.270</td>
<td>502.453</td>
<td>0.289</td>
</tr>
<tr>
<td>MINOR:</td>
<td>7.300</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MAJOR:</td>
<td>23.165</td>
<td>5.348</td>
<td>0.044</td>
</tr>
<tr>
<td>MINOR:</td>
<td>44.088</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLEXURE AND AXIAL TENSION:</td>
<td>C/S STRENGTH:</td>
<td>1.001</td>
<td>Cl. 13.9.1</td>
</tr>
<tr>
<td>MEM STR:</td>
<td>0.00E+00</td>
<td>2.65E+02</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>INTERMEDIATE:</td>
<td>TF:</td>
<td>0.00E+00</td>
<td>Tr:</td>
</tr>
<tr>
<td>STAAD SPACE</td>
<td>-- PAGE NO.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
V. CSA S16-14 - Beam Shear Capacity

Reference

Problem
The W530x82 beam has a maximum shear force, \( v = 540 \text{ kN} \) due to factored loads.

\[ L = 11.0 \text{ m} \]

The material used is G40.21 350W steel (\( F_y = 350 \text{ MPa} \)) (FYLD)

Calculations
Evaluate the slenderness effects of the beam web:

\[
\frac{h}{w} = \frac{528 - 2(13.3)}{9.5} = 52.8 < 439 \sqrt{\frac{F_y}{F_y}} = 439 \sqrt{\frac{534}{350}} = 54.2
\]

Therefore the shear capacity is calculated as:

\[
V_r = \phi A_w \times F_y = 0.9 \times 528 \times 9.5 \times 350 = 1,043 \text{ kN}
\]

Calculate Equivalent Concentrated Load
The given end moments and assumed distributed loads (which was determined to given an equivalent mid-span moment of the given example) do not result in a shear force of 540 kN. Therefore, an additional concentrated load is added near the first support.

\[
R_1 = \frac{w_f L}{2} + \frac{M_1 - M_2}{L} = \frac{40.17 \times 11.0}{2} + \frac{540 - 185}{11.0} = 253.2 \text{ kN}
\]

\[
P = 540 \text{ kN} - 253.2 \text{ kN} = 287 \text{ kN}
\]

The STAAD.Pro will have this load applied at 1 mm away from the left support.
### Table 517: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_r$ (kN)</td>
<td>1,043</td>
<td>1,042.826</td>
<td>negligible</td>
</tr>
</tbody>
</table>

#### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2014\CSA S16-14 - Beam Shear Capacity.STD is typically installed with the program.

```plaintext
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 ENFORCED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
1 MZ 540
2 MZ -185
MEMBER LOAD
1 UNI GY -40.033
1 CON GY -352.27 10.945
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
```
STAAD Output

STAAD.PRO CODE CHECKING - S16-14 (v1.0)
*******************************************************************************
ALL UNITS ARE - KN   MET (UNLESS OTHERWISE Noted)
-----------------------------  START OF DESIGN OUTPUT OF MEMBER 1
-----------------------------------------------
*MEMBER NO: 1  CRITICAL RATIO: 1.803(FAIL)  LOAD: 1
LOCATION (MET): 0.00  CONDITION: Cl. 13.8
SECTION: ST W530X82  (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 1.803(FAIL)  LOAD CASE: 1 LOCATION (MET): 0.00
CONDITION: Cl. 13.8
DESIGN FORCES:  Fx: 0.00(T)  Fy: 254.22  Fz: 0.00
Mx: 0.00E+00  My: 0.00E+00  Mz: 5.40E+02
UNIT: CM
SECTION PROPERTIES:  AZZ: 55.594  AYY: 48.896
CW: 1340255.500
SZZ: 1806.818  SYY: 194.258
IZZ: 47700.008  IYY: 2030.000
UNIT: NEW MM
MATERIAL PROPERTIES:  FYLD: 350.000  FU: 450.000
ACTUAL MEMBER LENGTH(MET): 11.000
PARAMETERS:  KZ: 1.000  KY: 1.000  NSF: 1.000  SLF: 1.000
SLENDERNESS:  ACTUAL SLENDERNESS RATIO: 250.172  LOAD: 1  LOC.
(MET): 0.000
ALLOWABLE SLENDERNESS RATIO: 300.000
SECTION CLASS:
COMPRESSION:  Class 4
FLEXURE  MAJOR  MINOR
SECTION:  Class 2  Class 2
FLANGE:  Class 2  Class 2
WEB:  Class 1
UNIT: KN MET
TENSION:  FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
YIELDING:  0.000  3307.500  0.000  Cl. 13.2
1  0.000
RUPTURE:  0.000  3543.750  0.000  Cl. 13.2
1  0.000
STAAD SPACE
--- PAGE NO.
6

STAAD.PRO CODE CHECKING - (S16-14) v1.0
*******************************************************************************
ALL UNITS ARE - KN   MET (UNLESS OTHERWISE Noted)
COMPRESSIOM:  FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
LOCATION(MET):
MAJOR:  0.000  2253.274  0.000  Cl. 13.3.5(a)
1  0.000
MINOR:  0.000  253.222  0.000  Cl. 13.3.5(a)
1  0.000
INTERMEDIATE:  Ag:(CM)  KL/r:  Fe:(N MM)  l  n:

MAJOR:  89.688     51.609     759.622     0.679     1.340
MINOR:  89.688     250.172     32.328     3.290     1.340
FLEX TOR BUCK: FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
LOCATION(MET):  0.000     854.388     0.000  Cl. 13.3.5(a)  1
0.000
INTERMEDIATE:  Fe:(N MM)  l
               125.187     1.672     1.340
SHEAR:  FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
LOCATION(MET):  MAJOR:  -538.417     1042.826     0.516  Cl. 13.4.1.1  1
           MINOR:  0.000     1050.727     0.000  AISC G2-1
1

INTERMEDIATE:  Aw:(CM)  Kv:  Ka:  Fcrl:(N MM)  Fcre:(N MM)
Fs:(N MM)
MAJOR:  48.896     5.349     0.048     237.748
MINOR:     5.594
UNIT:  KN MET
YIELDING: FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
LOCATION(MET):  MAJOR:  -5.40E+02     6.49E+02     0.832  Cl. 13.5(a)
           MINOR:  0.00E+00     9.54E+01     0.000  Cl. 13.5(a)
1

INTERMEDIATE:  Mp:  Se:  My:
MAJOR:  7.21E+02     0.00E+00     0.00E+00
MINOR:  1.06E+02     0.00E+00     0.00E+00
UNIT:  KN MET
LAT TOR BUCK: FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
LOCATION(MET):  MAJOR:  -5.40E+02     2.99E+02     1.803  Cl. 13.6(a)(ii
           MINOR:  0.00E+00     2.29     0.00     3.33E+02     0.00     0.00E+00
0.00     0.00
UNIT:  KN MET
INTERACTION: RATIO:  CRITERIA:  LOAD CASE:
LOCATION(MET):  FLEXURE AND AXIAL TENSION:
C/S STRENGTH:  0.832  Cl. 13.9.1  1  0.000
MEMBER STRENGTH:  1.803  Cl. 13.9.2  1  0.000
FLEXURE AND AXIAL COMPRESSION:
C/S STRENGTH:  0.707  Cl. 13.8.2  1  0.000
MEMBER STRENGTH:  0.707  Cl. 13.8.2  1  0.000
LTB STRENGTH:  1.533  Cl. 13.8.2  1  0.000
FLEX AND SHEAR:  1.422  Cl. 14.6(a)  1  0.000
BIAXIAL FLEX:  1.803  Cl. 13.8  1  0.000
STAAD SPACE -- PAGE NO.
7

INTERMEDIATE:
FLEXURE AND AXIAL TENSION:
C/S STRENGTH:  Tf:  0.00E+00  Tr:  3.31E+03  Mfz:  -5.40E+02  Mrz:
6.49E+02
FLEXURE AND AXIAL COMPRESSION:
Cf:  Mfz:  Mfy:  Cr:  Mrz:  Mry:  b:
V. CSA S16-14 - Select a Beam

Select a wide-flange shape for the given beam span.

Reference

Problem
A beam must be selected to span 11.0 m. The end moment of the span are:

\[ M_1 = -540 \text{ kN\cdot m} \]
\[ M_2 = -185 \text{ kN\cdot m} \]

And the maximum moment in the span is:

\[ M_3 = 256 \text{ kN\cdot m} \]

The beam is braced at 2.5 m from the center of each supporting column (thus \( L_b = 6.0 \text{ m} \)) (LT 6.0). The material used is G40.21 350W steel (\( F_y = 350 \text{ MPa} \)) (FYLD)

Calculations
Assume a Class 2 section:

\[ M_x = \phi M_p = \phi Z_x F_y \]

Thus,

\[ Z_x \geq \frac{540(10)^6}{0.9 \times 350} = 1, 714 \times (10)^3 \text{ mm}^3 \]

Try a W460x82:

\[ Z_x = 1,830(10)^3 \text{ mm}^3, tf = 16.0 \text{ mm}, bf = 191.0 \text{ mm}, tw = 9.9 \text{ mm}, D = 460.0 \text{ mm} \]

Evaluate the slenderness effects of the beam flanges:

\[ \frac{b}{2t} = \frac{191}{2 \times 16.0} = 5.7 < \frac{170}{\sqrt{350}} = 9.1 \]
Evaluate the slenderness effects of the beam web:

\[
\frac{h}{w} = \frac{460 - 2(16.0)}{9.9} = 43.2 < \frac{1.700}{\sqrt{350}} = 90.9
\]

The assumptions are valid and this beam is sufficient to support the given moments.

\[
M_r = 0.9 \times 1.830 \times 10^3 \times 350 \left[(10)^6\right] = 576.5 \text{ kN·m}
\]

Ratio = \(\frac{540}{576.5} = 0.937\)

**Comparison**

**Table 518: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section</td>
<td>W460x82 *</td>
<td>W460x82</td>
<td>none</td>
</tr>
<tr>
<td>(M_r) (kN·m)</td>
<td>576.5 **</td>
<td>576</td>
<td>negligible</td>
</tr>
<tr>
<td>Critical Ratio</td>
<td>0.937 **</td>
<td>0.937</td>
<td>none</td>
</tr>
</tbody>
</table>

**Notes:**

*The reference also tries a W530x82, which is stiffer for the same weight. STAAD.Pro selects the shallower beam.

** The reference does not calculate the resisting moment capacity nor the critical ratio for either section size, but they are evaluated here for completeness.

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2014\CSA S16-14 - Select a Beam.STD is typically installed with the program.

In order to model this beam in STAAD.Pro, we need to determine the distributed load which would result in the same mid-span moment on a beam with the given end moments. From the AISC Steel Construction Manual, p 3-222:

\[
M_3 = \frac{w_f L^2}{8} - \frac{(M_1 + M_2)^2}{2} + \frac{(M_1 - M_2)^2}{2w_f L^2}
\]

\[
256 = \frac{w_f (11.0)^2}{8} - \frac{(540 + 185)^2}{2} + \frac{(540 - 185)^2}{2w_f (11.0)^2}
\]

Solving this yields \(w_f = 40.033\) kN/m
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -40.033
JOINT LOAD
1 MZ 540
2 MZ -185
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
SELECT ALL
PERFORM ANALYSIS
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - S16-14 (v1.0)
******************************************************************************
ALL UNITS ARE - KN  MET (UNLESS OTHERWISE Noted)
----------------- START OF DESIGN OUTPUT OF MEMBER 1
-----------------
MEMBER NO: 1  CRITICAL RATIO: 0.832(PASS) LOAD: 1
LOCATION (MET): 0.00  CONDITION: Cl. 13.8
SECTION: ST W530X82  (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 0.832(PASS) LOAD CASE: 1 LOCATION (MET): 0.00
CONDITION: Cl. 13.8
DESIGN FORCES: Fx: 0.00(T) Fy: 252.45 Fz: 0.00

Verification Examples
V.09 Steel Design
Verification Examples
V.09 Steel Design

<table>
<thead>
<tr>
<th>Mx:</th>
<th>0.00E+00</th>
<th>My:</th>
<th>0.00E+00</th>
<th>Mz:</th>
<th>5.40E+02</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SZZ: 1806.818</td>
<td>SYY: 194.258</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>IZZ: 47700.008</td>
<td>IYY: 2030.000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>UNIT: NEW MM</td>
<td>MATERIAL PROPERTIES:</td>
<td>FYLD: 350.000</td>
<td>FU: 450.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ACTUAL MEMBER LENGTH(MET):</td>
<td>11.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PARAMETERS:</td>
<td>KZ: 1.000</td>
<td>KY: 1.000</td>
<td>NSF: 1.000</td>
<td>SLF: 1.000</td>
<td></td>
</tr>
<tr>
<td>SLENDERNESS:</td>
<td>ACTUAL SLENDERNESS RATIO: 250.172</td>
<td>LOAD: 1</td>
<td>LOC. (MET): 0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ALLOWABLE SLENDERNESS RATIO: 300.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SECTION CLASS: COMPRESSION: Class 4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLEXURE MAJOR MINOR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SECTION: Class 2 Class 2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLANGE: Class 2 Class 2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>WEB: Class 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>UNIT: KN MET</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TENSION: FORCE: 2253.274 RATIO: 0.000 CRITERIA: Cl. 13.3.5(a) LOAD CASE: 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MAJOR: 0.000 MINOR: 0.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>INTERMEDIATE: Ag:(CM) KL/r: 0.679 Fe:(N MM) l:n 1.340</td>
<td>MAJOR: 89.688 51.609 759.622</td>
<td>MINOR: 89.688 250.172 32.328</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLEX TOR BUCK: FORCE: 854.388 CAPACITY: 0.000 CRITERIA: Cl. 13.3.5(a) LOAD CASE: 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LOCATION(MET): 0.000 125.187</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>INTERMEDIATE: Fe:(N MM) 1 n: 1.672 1.340</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SHEAR: FORCE: 252.454 1042.826 0.242 CRITERIA: Cl. 13.4.1.1 LOAD CASE: 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LOCATION(MET): 1 0.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>INTERMEDIATE: Aw:(CM) Kv: 1 Ka: 0.048 237.748</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fs:(N MM) MAJOR: 48.896 5.349 345.654</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MINOR: 55.594 231.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
UNIT: KN MET
YIELDING: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION (MET):
MAJOR:  -5.40E+02  6.49E+02   0.832   Cl. 13.5(a)
1  0.000
MINOR:  0.00E+00   9.54E+01   0.000   Cl. 13.5(a)
1  0.000
INTERMEDIATE:  Mp:  Se:  My:
MAJOR:  7.21E+02  0.00E+00  0.00E+00
MINOR:  1.06E+02  0.00E+00  0.00E+00
UNIT: KN MET
INTERACTION: RATIO: CRITERIA: LOAD CASE:
LOCATION (MET):
FLEXURE AND AXIAL TENSION:
C/S STRENGTH: 0.832 Cl. 13.9.1 1 0.000
FLEXURE AND AXIAL COMPRESSION:
C/S STRENGTH: 0.707 Cl. 13.8.2 1 0.000
MEMBER STRENGTH: 0.707 Cl. 13.8.2 1 0.000
FLEX AND SHEAR: 0.715 Cl. 14.6(a) 1 0.000
BIAXIAL FLEX: 0.832 Cl. 13.8 1 0.000
INTERMEDIATE:
FLEXURE AND AXIAL TENSION:
C/S STRENGTH: Tf: 0.00E+00  Tr: 3.31E+03 Mfz: -5.40E+02 Mrz: 6.49E+02
STAAD SPACE
-- PAGE NO.
7
STAAD.PRO CODE CHECKING - (S16-14) v1.0
ALL UNITS ARE - KN MET (UNLESS OTHERWISE NOTED)
FLEXURE AND AXIAL COMPRESSION:
Cf:  Mfz:  Cr:  Mrz:  Mry:  b :
Cez:  Cey:  w 1z:  w 1y:  U1z:  U1y:
C/S STR: 0.00E+00 -5.40E+02  0.00E+00  2.83E+03  6.49E+02  9.54E+01  0.85  7.98E +03  3.39E+02  0.74  0.60  1.00  1.00
MEM STR: 0.00E+00 -5.40E+02  0.00E+00  2.53E+02  6.49E+02  9.54E+01  0.85  7.98E +03  3.39E+02  0.60  0.74  1.00  1.00
FLEX SHEAR: Mfz: -5.40E+02 Mrz: 6.49E+02 Vfy: 1.04E+03
BIAX FLEX: Mfz: -5.40E+02 Mrz: 6.49E+02 Myf: 0.00E+00 Mrv: 9.54E+01
END OF DESIGN OUTPUT OF MEMBER 1
-------------------------
***NOTE: OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF
ANALYTICAL MEMBERS
45. SELECT ALL
STEEL DESIGN
STAAD SPACE
-- PAGE NO.
8
STAAD.PRO MEMBER SELECTION - S16-14 (v1.0)
ALL UNITS ARE - KN MET (UNLESS OTHERWISE NOTED)
------------------------- START OF DESIGN OUTPUT OF MEMBER 1
-------------------------
MEMBER NO: 1 CRITICAL RATIO: 0.937(PASS) LOAD: 1
LOCATION (MET): 0.00 CONDITION: Cl. 13.8
SECTION: ST W460x82 (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 0.937 (PASS)  LOAD CASE:  1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8

DESIGN FORCES:
Fx: 0.00 (T)  Fy: 252.45  Fz: 0.00
Mx: 0.00E+00  My: 0.00E+00  Mz: 5.40E+02

UNIT: CM

SECTION PROPERTIES:
AZZ: 61.120  AYY: 43.956  CW: 1340255.500
SZZ: 1608.696  SYY: 194.764
IZZ: 37000.004  IYY: 1860.000

UNIT: NEW MM

MATERIAL PROPERTIES:
FYLD: 350.000  FU: 450.000

ACTUAL MEMBER LENGTH (MET): 11.000

PARAMETERS:
KZ: 1.000  KY: 1.000  NSF: 1.000  SLF: 1.000

SLENDERNESS:
ACTUAL SLENDERNESS RATIO: 260.107  LOAD: 1  LOC. (MET): 0.000
ALLOWABLE SLENDERNESS RATIO: 300.000

SECTION CLASS:
COMPRESSION: Class 4
FLEXURE:
MAJOR: Class 1  MINOR: Class 1
SECTION: Class 1  Class 1
FLANGE: Class 1  Class 1
WEB: Class 1
UNIT: KN MET

YIELDING:
MAJOR: 0.000  3276.000  0.000  Cl. 13.2
MINOR: 0.000  965.250  0.000  Cl. 13.2

RUPTURE:
MAJOR: 0.000  2261.248  0.000  Cl. 13.3.5(a)
MINOR: 0.000  253.374  0.000  Cl. 13.3.5(a)

INTERMEDIATE:
Ag:(CM)  KL/r:  Fe:(N MM)  l  n:
MAJOR: 96.728  58.319  594.890  0.767  1.340
MINOR: 96.728  260.107  29.905  3.421  1.340

FLEX TORSION BUCKLING:
FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
MAJOR: 0.000  1279.603  0.000  Cl. 13.3.5(a) 1
MINOR: 0.000  1155.168  0.000  AISC G2-1

INTERMEDIATE:
Aw:(CM)  Kv:  Ka:  Fcri:(N MM)  Fcre:(N MM)

SHEAR:
FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:
MAJOR: 252.454  946.776  0.267  Cl. 13.4.1.1
MINOR: 0.000  1155.168  0.000  AISC G2-1

INTERMEDIATE:
Fv:(N MM)  1  n:

STAAD SPACE
### V. CSA S16-14 - Shear Capacity Combined Stresses

**Reference**

Problem
The W250x73 column has the following concentrated loads
\[ C_f = 900 \text{ kN} \]
And a uniform moment of:
\[ M_{fx} = 180 \text{ kN m} \]
The material used is G40.21 350W steel \( (F_y = 350 \text{ MPa}) \) \( \text{(FYLD)} \). The beam is braced against lateral-torsional buckling only at the ends \( \text{(LAT \ 0, default)} \).

Calculations

Axial Capacity
Determine the buckling load of the column:
\[
\left( \frac{KL}{r} \right)_x = \frac{1.0(3,600)}{110} = 32.7
\]
\[
\left( \frac{KL}{r} \right)_y = \frac{1.0(3,600)}{64.6} = 55.7
\]
The largest slenderness ratio controls:
\[
F_{ex} = \frac{n^2E}{(KL/r)_x^2} = \frac{n^2(200,000)}{(55.7)^2} = 635.6 \text{ kN}
\]
\[
\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{635.6}} = 0.742
\]
\[
C_r = \phi AF_y (1 + \lambda^{2n})^{-1/n} = 0.90 \times 9, 280 \times 350[1 + 0.742^{(2 \times 1.34)}]^{-1/1.34} = 2, 216 \times (10)^6 \text{ N} = 2, 216 \text{ kN}
\]

Moment Modifier
Uniform moment:
\[
\omega_1 = 1.0
\]
Calculate the elastic buckling load for the bending axis:
\[
C_e = \frac{n^2 \times 200,000 \times 113 \times 10^6}{(1.0 \times 3,600)^2} = 17, 211 \text{ kN}
\]
\[
U_{1x} = \frac{1.0}{1 - \frac{900}{17,211}} = 1.06
\]

Bending Capacity
\[
\omega_2 = 1.0
\]
\[
EI_yGJ = 200,000 \times 38.8 \times 76.92 \times 575 = 343.2 \times 10^{21} \text{ MPa}^2 \text{ mm}^8
\]
\[
\left( \frac{nE}{L} \right)^2 I_y C_w = \left( \frac{n \times 200,000}{3,600} \right)^2 38.8 \times 553 = 653.6 \times 10^{21} \text{ MPa}^2 \cdot \text{ mm}^8
\]
\[ M_u = \frac{\omega^2}{L^2} EI_y GJ + \left(\frac{nK}{L}\right)^2 I_y C_w = \frac{1.0 \times 934.2 \times 10^{21} + 653.6 \times 10^{21}}{3,600^2} = 871 \times 10^6 \text{ N mm} = 871 \text{ kN m} \]

Plastic moment capacity:
\[ M_p = Z_x F_y = 985 \times 10^3 \times 350 = 345 \times 10^6 \text{ N mm} = 345 \text{ kN m} \]
\[ M_u = 871 \text{ kN m} > 0.67 \times M_p = 0.67 \times 345 = 231 \text{ kN m} \]

Therefore, the moment capacity is calculated as:
\[ M_r = 1.15 \phi M_p \left(1 - \frac{0.28 M_p}{M_u}\right) = 1.15 \times 0.9 \times 345 \left(1 - \frac{0.28 \times 345}{871}\right) = 317 \text{ kN m} \]

Check the capacity based on the strength cross-section:
\[ M_{rx} = 0.9 \times 985 \times 10^3 \times 350 = 310 \text{ kN m} \]

This governs capacity.

**Combined Stress Ratio**
\[
\frac{900}{2,216 \times 0.85 \times 1.06 \times 180} = 0.406 + 0.523 = 0.929 < 1.0
\]

**Comparison**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(KL/r)_x</td>
<td>33</td>
<td>32.624</td>
<td>negligible</td>
</tr>
<tr>
<td>(KL/r)_y</td>
<td>56</td>
<td>55.675</td>
<td>negligible</td>
</tr>
<tr>
<td>C_r (kN)</td>
<td>2,216</td>
<td>2,220</td>
<td>negligible</td>
</tr>
<tr>
<td>C_e (kN)</td>
<td>17,211</td>
<td>17,200</td>
<td>negligible</td>
</tr>
<tr>
<td>Stress Ratio</td>
<td>0.929</td>
<td>0.926</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**Note:** Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

**STAAD Input**

The file
C:\Users\Public\Documents\STAAD.Pro CONNEC Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2014\CSA S16-14 - Shear Capacity Combined Stresses.STD is typically installed with the program.

**STAAD SPACE**
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.00e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
*MEMBER PROPERTY CANADIAN
*1 TABLE ST W250X73
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
2 ENFORCED BUT FY MX MZ
*2 ENFORCED BUT FX FZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
1 FY 900 MZ 180
2 FY -900 MZ -180
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN
SSY 1 ALL
SSZ 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - S16-14 (v1.0)
*****************************************************************************
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
------------------ START OF DESIGN OUTPUT OF MEMBER 1 ------------------
MEMBER NO: 1 CRITICAL RATIO: 0.926(PASS) LOAD: 1
LOCATION (MET): 0.00 CONDITION: Cl. 13.8.2
SECTION: ST W250X73 (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 0.926(PASS) LOAD CASE: 1 LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
DESIGN FORCES: Fx: 900.00(C)  Fy: 0.00  Fz: 0.00
Mx: 0.00E+00  My: 0.00E+00  Mz: 1.80E+02
UNIT: CM
SECTION PROPERTIES: AZZ: 72.136  AYY: 20.537  CW: 552900.312
SZZ: 893.281  SYY: 305.512  IZZ: 11300.001  IYY: 3880.000
UNIT: NEW MM
MATERIAL PROPERTIES: FYLD: 350.000  FU: 450.000
ACTUAL MEMBER LENGTH(MET): 3.600
PARAMETERS: KZ: 1.000  KY: 1.000  NSF: 1.000  SLF: 1.000
SLENDERNESS: ACTUAL SLENDERNESS RATIO: 55.675  LOAD: 1  LOC. (MET): 0.000
ALLOWABLE SLENDERNESS RATIO: 200.000
SECTION CLASS:
COMPRESSION: Class 1
FLEXURE  MAJOR  MINOR
SECTION: Class 2  Class 2
FLANGE: Class 2  Class 2
WEB: Class 1
UNIT: KN MET
TENSION: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
YIELDING: 0.000  2923.200  0.000  Cl. 13.2
1 0.000
RUPTURE: 0.000  3132.000  0.000  Cl. 13.2
1 0.000
STAAD SPACE -- PAGE NO. 6

**********

UNIT: KN MET
YIELDING: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
MAJOR: -1.80E+02  3.10E+02  0.580  Cl. 13.5(a)
MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1        0.000
INTERMEDIATE:   Mp:        Se:         My:
MAJOR:        3.45E+02    0.00E+00    0.00E+00
MINOR:        1.62E+02    0.00E+00    0.00E+00
UNIT: KN MET
LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
MAJOR:       -1.80E+02    3.17E+02      0.567    Cl. 13.6(a)(i)
1        0.000
INTERMEDIATE:  Iyc(CM):   w 2:     w 3:       b x:     Mu:     rt:
Myr:     Lu:     Ly:
MAJOR:        0.00E+00  1.00     0.00     0.00   8.71E+02 0.00   0.00E+00
0.00   0.00
UNIT: KN MET
INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
FLEXURE AND AXIAL TENSION:
C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:  -1.80E+02  Mrz:
3.10E+02
STAAD SPACE
-- PAGE NO. 7
***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF
ANALYTICAL MEMBERS
V. CSA S16-14 - Short Column Compression

Determine the factored compressive resistance of a short column.

Reference
Example 4.1

Problem
A W250x73 section is used for a pedestal with a height of 1.1 m.
The material used is G40.21 350W steel ($F_y = 350 \text{ MPa}$) (FYLD)

Calculations
The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling
Evaluate the slenderness effects of the column web:

$$\frac{b}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity
Assume $K = 1.0$,

$$\frac{L}{r_x} = \frac{1,100}{110} = 10$$

$$\frac{L}{r_y} = \frac{1,100}{64.6} = 17$$

The largest slenderness ratio controls, thus the slenderness factor for buckling about the minor axis:

$$\lambda = \left(\frac{KL}{r}\right)_{\text{max}} \sqrt{\frac{F_y}{\pi^2 E}} = 17 \sqrt{\frac{350}{\pi^2 200,000}} = 0.227$$

When $n = 1.34$ (Group 2 W shape),

$$C_r = \phi A F_y (1 + \lambda^{2n})^{-1/n} = 0.90 \times 9, 280 \times 350 \left[1 + 0.227^{(2 \times 1.34)}\right]^{-1/1.34} = 2.883 \times (10)^6 \text{ N} = 2, 883 \text{ kN}$$
### Table 520: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/r, major</td>
<td>10</td>
<td>9.968</td>
<td>negligible</td>
</tr>
<tr>
<td>KL/r, minor</td>
<td>17</td>
<td>17.012</td>
<td>negligible</td>
</tr>
<tr>
<td>$\lambda$, critical</td>
<td>0.227</td>
<td>0.224</td>
<td>1.3%</td>
</tr>
<tr>
<td>$C_r$(kN)</td>
<td>2,883</td>
<td>2,884.358</td>
<td>negligible</td>
</tr>
</tbody>
</table>

#### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2014\CSA S16-14 - Short Column Compression.STD is typically installed with the program.

```plaintext
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1.1 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
*1 TABLE ST L203x203x25
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
*2 FY -100
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
```
STAAAD Output

------------------------ START OF DESIGN OUTPUT OF MEMBER 1 ------------------------

MEMBER NO: 1  CRITICAL RATIO: 0.000(PASS)  LOAD: 1
LOCATION (MET): 0.00  CONDITION: Cl. 13.8.2
SECTION: ST W250X73  (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 0.000(PASS)  LOAD CASE: 1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
DESIGN FORCES: Fx: 0.00(C)  Fy: 0.00  Fz: 0.00
Mx: 0.00E+00  My: 0.00E+00  Mz: 0.00E+00
UNIT: CM
SECTION PROPERTIES: AZZ: 72.136  AYY: 20.537  CW: 552900.312
SZZ: 893.281  SYY: 305.512  IZZ: 11300.001  IYY: 3880.000
UNIT: NEW MM
MATERIAL PROPERTIES: FYLD: 350.000  FU: 450.000
ACTUAL MEMBER LENGTH(MET): 1.100
PARAMETERS:
KZ: 1.000  KY: 1.000  NSF: 1.000  SLF: 1.000
SLENDERNESS: ACTUAL SLENDERNESS RATIO: 17.012  LOAD: 1  LOC. (MET): 0.00
ALLOWABLE SLENDERNESS RATIO: 200.000
SECTION CLASS:
COMPRESSION: Class 1
FLEXURE MAJOR  MINOR
SECTION: Class 2  Class 2
FLANGE: Class 2  Class 2
WEB: Class 1
UNIT: KN MET
TENSION: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
YIELDING: 0.000  2923.200  0.000  Cl. 13.2
1 0.000  3132.000  0.000  Cl. 13.2
1 0.000  STAAD SPACE

STAAD Pro CODE CHECKING - (S16-14) v1.0
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
COMPRESSION: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
MAJOR: 0.001  2913.816  0.000  Cl. 13.3
1 0.000  2884.358  0.000  Cl. 13.3
1 0.000  -- PAGE NO. 6
### Verification Examples

**V.09 Steel Design**

<table>
<thead>
<tr>
<th>LOCATION (MET):</th>
<th>MAJOR:</th>
<th>MINOR:</th>
<th>INTERMEDIATE:</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION (MET):</td>
<td>MAJOR:</td>
<td>MINOR:</td>
<td>INTERMEDIATE:</td>
</tr>
<tr>
<td>LOCATION (MET):</td>
<td>MAJOR:</td>
<td>MINOR:</td>
<td>INTERMEDIATE:</td>
</tr>
<tr>
<td>LOCATION (MET):</td>
<td>MAJOR:</td>
<td>MINOR:</td>
<td>INTERMEDIATE:</td>
</tr>
</tbody>
</table>

### Units

- **KN MET**

### Load Cases

- **Cl. 13.4.1.1**
- **Cl. 13.5(a)**
- **Cl. 13.6(a)(i)**
- **Cl. 13.8.2**
- **Cl. 13.9.1**
- **Cl. 13.8**

### Criteria

- **C/S STR: 1.00E-03**
- **C/S ST: 1.00E-03**
- **C/S STR: 1.00E-03**
- **C/S STR: 1.00E-03**
- **C/S STR: 1.00E-03**
- **C/S STR: 1.00E-03**

### Summary

- **STAAD SPACE**

---

**ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)**

### Additional Notes

- **STAAD.PRO CODE CHECKING - (S16-14) v1.0**

---

**STAAD.Pro User Manual**
V. CSA S16-14 - Slender Column Compression

Determine the factored compressive resistance of a slender column.

Reference

Problem
A W250x73 section is used for a pedestal with a height of 11.0 m. The material used is G40.21 350W steel \((F_y = 350 \text{ MPa})\) (FYLD)

Calculations
The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling
Evaluate the slenderness effects of the column web:
\[
\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8
\]
Evaluate the slenderness effects of the column flanges:
\[
\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7
\]
Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity
Assume \(K = 1.0\),
\[
\frac{L}{r_x} = \frac{11,000}{110} = 100
\]
\[ \frac{L}{r_y} = \frac{11,000}{64.6} = 170 \]

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

\[ F_e = \frac{\pi^2 \times 200,000}{(1.0 \times 170)^2} = 68.3 \text{kN} \]

\[ \lambda = \sqrt{\frac{F_y}{F_e}} = \sqrt{\frac{350}{68.3}} = 2.26 \]

When \( n = 1.34 \) (Group 2 W shape),

\[ C_r = \phi A F_y (1 + \lambda^{2n})^{-1/n} = 0.90 \times 9, 280 \times 350 \left[1 + 2.26^{(2 \times 1.34)}\right]^{-1/1.34} = 0.529 \times (10)^6 \text{N} = 529 \text{kN} \]

**Comparison**

**Table 521: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/r, major</td>
<td>100</td>
<td>99.684</td>
<td>negligible</td>
</tr>
<tr>
<td>KL/r, minor</td>
<td>170</td>
<td>170.118</td>
<td>negligible</td>
</tr>
<tr>
<td>( \lambda ), critical</td>
<td>2.26</td>
<td>2.237</td>
<td>1.0%</td>
</tr>
<tr>
<td>( C_r )(kN)</td>
<td>529</td>
<td>538.161</td>
<td>1.7%</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2014\CSA S16-14 - Slender Column Compression.STD is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Apr-14
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 11 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
```
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

1

*******************************************************************************
* *
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 23: 8:28 *
* *
* Licensed to: Bentley Systems Inc *
*******************************************************************************
1. STAAD SPACE
INPUT FILE: CSA S16-14 - Slender Column Compression.STD
2. START JOB INFORMATION
3. ENGINEER DATE 22-APR-14
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER KN
7. JOINT COORDINATES
8. 1 0 0 0; 2 0 11 0
9. MEMBER INCIDENCES
10. 1 1 2
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 2.05E+08
14. POISSON 0.3
15. DENSITY 76.92
16. ALPHA 1.2E-05
17. DAMP 0.03
18. TYPE STEEL
19. *STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
20. G 7.692E+07
21. END DEFINE MATERIAL
22. MEMBER PROPERTY CANADIAN
23. 1 TABLE ST W250X73
24. CONSTANTS
25. MATERIAL STEEL ALL
26. SUPPORTS
27. 1 FIXED
28. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
29. JOINT LOAD
30. 2 FY -0.001
31. PERFORM ANALYSIS
STAAD SPACE

2

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          2   NUMBER OF MEMBERS       1
NUMBER OF PLATES          0   NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0   NUMBER OF SUPPORTS      1
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       6
TOTAL LOAD COMBINATION CASES =     0 SO FAR.

32. PRINT MEMBER PROPERTIES ALL
MEMBER PROPERTIES.
MEMBER   PROFILE              AX/          IZ/          IY/        IX/        AY           AZ           SZ         SY
1  ST  W250X73           92.80     11300.00      3880.00       57.50
21.76        48.09       893.28      305.51

************ END OF DATA FROM INTERNAL STORAGE ************
33. PRINT SUPPORT REACTION ALL
SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
1   1      0.00      0.00      0.00      0.00      0.00      0.00

************** END OF LATEST ANALYSIS RESULT **************
34. PARAMETER 1
35. CODE CANADIAN 2014
36. FYLD 350000 ALL
37. FU 450000 ALL
38. TRACK 2 ALL
39. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE

3

MEMBER PROPERTIES. UNIT - CM
-----------------------------
MEMB   PROFILE              AX/          IZ/          IY/        IX//       AY           AZ           SZ         SY
1  ST  W250X73           92.80     11300.00      3880.00       57.50
21.76        48.09       893.28      305.51

************ END OF DATA FROM INTERNAL STORAGE ************
33. PRINT SUPPORT REACTION ALL
SUPPORT REACTION ALL

************** END OF LATEST ANALYSIS RESULT **************
34. PARAMETER 1
35. CODE CANADIAN 2014
36. FYLD 350000 ALL
37. FU 450000 ALL
38. TRACK 2 ALL
39. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE

4

SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
-----------------------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
1   1      0.00      0.00      0.00      0.00      0.00      0.00

************** END OF LATEST ANALYSIS RESULT **************
34. PARAMETER 1
35. CODE CANADIAN 2014
36. FYLD 350000 ALL
37. FU 450000 ALL
38. TRACK 2 ALL
39. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE

5

STAAD.PRO CODE CHECKING - S16-14 (v1.0)
ALL UNITS ARE - KN MET (UNLESS OTHERWISE Noted)
----------------------------- START OF DESIGN OUTPUT OF MEMBER 1
-----------------------------
MEMBER NO:  1 CRITICAL RATIO:  0.000(PASS) LOAD:  1
LOCATION (MET):  0.00 CONDITION: Cl. 13.8.2
### STAAD Space - Verification Examples

**3746**

**STAAD.Pro User Manual**

---

#### Verification Examples

**V.09 Steel Design**

---

```plaintext
SECTION: ST W250X73  (CANADIAN SECTIONS)
UNIT: KN MET
STRENGTH CHECKS:
CRITICAL RATIO: 0.000(PASS)  LOAD CASE: 1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
DESIGN FORCES:  
Fx: 0.00(C)  Fy: 0.00  Fz: 0.00  
Mx: 0.00E+00  My: 0.00E+00  Mz: 0.00E+00
UNIT: CM
SECTION PROPERTIES:  
UNIT: NEW MM
MATERIAL PROPERTIES:  
FYLD: 350.000  FU: 450.000
ACTUAL MEMBER LENGTH(MET): 11.000
PARAMETERS:  
KZ: 1.000  KY: 1.000  NSF: 1.000  SLF: 1.000
SLENDERNESS:  
ACTUAL SLENDERNESS RATIO: 170.118  LOAD: 1  LOC. (MET): 0.000
ALLOWABLE SLENDERNESS RATIO: 200.000
SECTION CLASS:  
COMPRESSION: Class 1  FLEXURE: MAJOR MINOR
SECTION: Class 2  Class 2  FLANGE: Class 2  Class 2
WEB: Class 1
UNIT: KN MET
TENSION:  
MAJOR: 0.001 1266.716 0.000  Cl. 13.3 1
MINOR: 0.001 538.161 0.000  Cl. 13.3 1
INTERMEDIATE: Ag:(CM)  KL/r:  Fe:(N MM)  l:  n:
MAJOR: 92.800 99.684 203.610 1.311 1.340
MINOR: 92.800 170.118 69.912 2.237 1.340
SHEAR:  
MAJOR: 0.000 452.349 0.000  Cl. 13.4.1.1 1
MINOR: 0.000 1363.370 0.000  AISC G2-1 1
Fs:(N MM)  MAJOR: 20.537 5.342 0.023 480.151
MINOR: 1400.819 231.000 72.136
```

---

**STAAD.Pro Code Checking - (S16-14) v1.0**

---

**Verification Examples**

**V.09 Steel Design**

---

**STAAD.Pro User Manual**
<table>
<thead>
<tr>
<th>Location(MET):</th>
<th>Major:</th>
<th>Minor:</th>
<th>Intermediate:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FORCE:</td>
<td>CAPACITY:</td>
<td>RATIO:</td>
</tr>
<tr>
<td>Major:</td>
<td>0.00E+00</td>
<td>3.10E+02</td>
<td>0.000</td>
</tr>
<tr>
<td>Minor:</td>
<td>0.00E+00</td>
<td>1.46E+02</td>
<td>0.000</td>
</tr>
<tr>
<td>Intermediate:</td>
<td>Mp:</td>
<td>Se:</td>
<td>My:</td>
</tr>
<tr>
<td>Major:</td>
<td>3.45E+02</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Minor:</td>
<td>1.62E+02</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Lat Tor Buck:</td>
<td>FORCE:</td>
<td>CAPACITY:</td>
<td>RATIO:</td>
</tr>
<tr>
<td>Major:</td>
<td>0.00E+00</td>
<td>2.83E+02</td>
<td>0.000</td>
</tr>
<tr>
<td>Intermediate:</td>
<td>Iyc(CM):</td>
<td>w 2:</td>
<td>w 3:</td>
</tr>
<tr>
<td>Major:</td>
<td>0.00E+00</td>
<td>2.50</td>
<td>0.00</td>
</tr>
<tr>
<td>Unit: KN MET</td>
<td>INTERACTION:</td>
<td>RATIO:</td>
<td>CRITERIA:</td>
</tr>
<tr>
<td>Location(MET):</td>
<td>FLEXURE AND AXIAL TENSION:</td>
<td>C/S STRENGTH:</td>
<td>Cl. 13.9.1</td>
</tr>
<tr>
<td></td>
<td>FLEXURE AND AXIAL COMPRESSION:</td>
<td>C/S STRENGTH:</td>
<td>Cl. 13.8.2</td>
</tr>
<tr>
<td></td>
<td>MEMBER STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.2</td>
</tr>
<tr>
<td></td>
<td>LTB STRENGTH:</td>
<td>0.000</td>
<td>Cl. 13.8.2</td>
</tr>
<tr>
<td></td>
<td>FLEX AND SHEAR:</td>
<td>0.000</td>
<td>Cl. 14.6(a)</td>
</tr>
<tr>
<td></td>
<td>BIAXIAL FLEX:</td>
<td>0.000</td>
<td>Cl. 13.8</td>
</tr>
<tr>
<td></td>
<td>INTERMEDIATE:</td>
<td>FLEXURE AND AXIAL TENSION:</td>
<td>C/S STRENGTH:</td>
</tr>
<tr>
<td></td>
<td>3.10E+02</td>
<td>2.92E+03</td>
<td>0.00E+00</td>
</tr>
<tr>
<td></td>
<td>3.10E+02</td>
<td>1.46E+02</td>
<td>0.85</td>
</tr>
<tr>
<td></td>
<td>3.10E+02</td>
<td>1.46E+02</td>
<td>0.85</td>
</tr>
<tr>
<td></td>
<td>3.10E+02</td>
<td>2.83E+02</td>
<td>1.46E+02</td>
</tr>
<tr>
<td></td>
<td>3.10E+02</td>
<td>2.83E+02</td>
<td>1.46E+02</td>
</tr>
<tr>
<td></td>
<td>3.10E+02</td>
<td>2.83E+02</td>
<td>1.46E+02</td>
</tr>
<tr>
<td></td>
<td>3.10E+02</td>
<td>2.83E+02</td>
<td>1.46E+02</td>
</tr>
<tr>
<td></td>
<td>FLEX AND SHEAR:</td>
<td>Mfz:</td>
<td>Mrz:</td>
</tr>
<tr>
<td></td>
<td>0.00E+00</td>
<td>2.83E+02</td>
<td>0.00E+00</td>
</tr>
<tr>
<td></td>
<td>BIAX FLEX:</td>
<td>Mfz:</td>
<td>Mrz:</td>
</tr>
<tr>
<td></td>
<td>0.00E+00</td>
<td>2.83E+02</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

***NOTE: OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL MEMBERS
40. FINISH

******** END OF THE STAAD.Pro RUN ********
V. CSA S16-14 - Wide Flange Capacity Combined Stresses

Check the capacity of a wide-flange column under combined stresses.

Reference

Problem
The W250x73 column has the following concentrated loads applied at the top of the column:

\[ C_f = 900 \text{ kN} \]
\[ M_{fx} = 180 \text{ kN} \cdot \text{m} \]

The material used is G40.21 350W steel \( (F_y = 350 \text{ MPa}) \) (FYLD). The column is assumed to be braced laterally against lateral-torsional buckling (LAT 1).

\[ L = 3.6 \text{ m} \]

Calculations

Local Element Buckling

Evaluate the slenderness effects of the column web:

\[ \frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{350} = 35.8 \]

Evaluate the slenderness effects of the column flanges:

\[ \frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7 \]

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.
Axial Capacity

Determine the buckling load of the column, assuming that weak axis bucking will not control due to bracing:

\[
\left( KL \right)_x = \frac{1.0 \times 3,600}{110} = 32.7
\]

The largest slenderness ratio controls:

\[
F_{ex} = \frac{n^2 E}{KL_x^2} = \frac{n^2 (200,000)}{(32.7)^2} = 1,847 \text{ kN}
\]

\[
\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{1,847}} = 0.435
\]

\[
C_r = \phi A F_y \left( 1 + \lambda^{2n} \right)^{-1/n} = 0.90 \times 9,280 \times 350 \left[ 1 + 0.435 \left( 2 \times 1.34 \right) \right]^{-1/1.34} = 2,716 \times (10)^6 \text{ N} = 2,716 \text{ kN}
\]

Moment Modifier

\[
\kappa = -0/180 = 0
\]

\[
\omega_1 = 0.6 - 0.4\kappa = 0.6 \leq 1.0
\]

Calculate the elastic buckling load for the bending axis:

\[
C_e = \frac{n^2 \times 200,000 \times 113 \times 10^6}{(1.0 \times 3,600)^2} = 17,211 \text{ kN}
\]

\[
U_{1x} = \frac{0.6}{1 - \left( \frac{900}{17,920} \right)} = 0.633
\]

Bending Capacity

From a previous example, the W250x73 is a Class 2 section. It is laterally supported along its length, thus:

\[
M_{rx} = 0.9 \times 985 \times 10^3 \times 350 = 310 \text{ kN} \cdot \text{m}
\]

Combined Stress Ratio

\[
\frac{900}{2,716} + \frac{0.85 \times 0.633 \times 180}{310} = 0.331 + 0.312 = 0.644 < 1.0
\]

This is the check for overall member strength. However, there is another case that actually controls in this instance: checking for cross sectional strength.

\[
C_r = 0.9 \times 9,280 \times 350 = 2,923 \text{ kN}
\]

\[
U_{1x} = 1.0
\]

\[
\frac{900}{2,923} + \frac{0.85 \times 1.0 \times 180}{310} = 0.308 + 0.494 = 0.801 < 1.0
\]
Comparison

Table 522: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(KL/r)_x</td>
<td>33</td>
<td>32.624</td>
<td>negligible</td>
</tr>
<tr>
<td>(KL/r)_y</td>
<td>56</td>
<td>55.675</td>
<td>negligible</td>
</tr>
<tr>
<td>C_r (kN)</td>
<td>2,716</td>
<td>2,715.935</td>
<td>negligible</td>
</tr>
<tr>
<td>C_e (kN)</td>
<td>17,211</td>
<td>17,600</td>
<td>2.3%</td>
</tr>
<tr>
<td>Stress Ratio</td>
<td>0.644</td>
<td>0.643</td>
<td>none</td>
</tr>
<tr>
<td>Stress Ratio (Critical)</td>
<td>0.801</td>
<td>0.801</td>
<td>none</td>
</tr>
</tbody>
</table>

**Note:** Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2014\CSA S16-14 - Wide Flange Capacity Combined Stresses.STD is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
```
### Staad Output

STAAD.PRO CODE CHECKING - S16-14 (v1.0)

ALL UNITS ARE - KN   MET (UNLESS OTHERWISE Noted)

--------------- START OF DESIGN OUTPUT OF MEMBER 1

MEMBER NO: 1  CRITICAL RATIO: 0.896(PASS)  LOAD: 1
LOCATION (MET): 3.60  CONDITION: Cl. 13.8.2
SECTION: ST W250X73  (CANADIAN SECTIONS)
UNIT: KN MET

STRENGTH CHECKS:
CRITICAL RATIO: 0.896(PASS)  LOAD CASE: 1  LOCATION (MET): 3.60
CONDITION: Cl. 13.8.2
DESIGN FORCES: Fx: 900.00(C)  Fy: 50.00  Fz: 0.00
My: 0.00E+00  Mz: -1.80E+02
UNIT: CM
SECTION PROPERTIES: AZZ: 72.136 AYY: 20.537 CW:
552900.312
SZZ: 893.281 SYY: 305.512 IZZ: 11300.001 IYY: 3880.00
UNIT: NEW MM
MATERIAL PROPERTIES: FYLD: 350.000  FU: 450.000
ACTUAL MEMBER LENGTH(MET): 3.600
PARAMETERS: KZ: 1.000  KY: 1.000  NSF: 1.000  SLF: 1.000
SLENDERNESS: ACTUAL SLENDERNESS RATIO: 55.675 LOAD: 1 LOC.
(MET): 0.000  ALLOWABLE SLENDERNESS RATIO: 200.000
SECTION CLASS: COMPRESSION: Class 1
FLEXURE  MAJOR  MINOR
SECTION: Class 2  Class 2
FLANGE: Class 2  Class 2
WEB: Class 1
UNIT: KN MET
TENSION: FORCE: CAPACITY: RATIO: CRITERIA: LOAD CASE:
LOCATION(MET):
YIELDING:  0.000  2923.200  0.000  Cl. 13.2
1 0.000
RUPTURE:  0.000  3132.000  0.000  Cl. 13.2
1 0.000

--- PAGE NO. 6 ---

STAAD SPACE

STAAD.PRO CODE CHECKING - (S16-14) v1.0

*************************************************

ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)

COMPRESSION:  FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:

LOCATION(MET):
MAJOR:       900.000  2715.935  0.331  Cl. 13.3
1 0.000
MINOR:       900.000  2233.940  0.403  Cl. 13.3
1 0.000

INTERMEDIATE:  Ag:(CM)  KL/r:  Fe:(N MM)  l  n:
MAJOR:        92.800  32.624  1900.987  0.429  1.340
MINOR:        92.800  55.675  652.728  0.732  1.340

SHEAR:  FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:

LOCATION(MET):
MAJOR:       50.000  452.349  0.111  Cl. 13.4.1.1
1 0.000
MINOR:       0.000  1363.370  0.000  AISC G2-1
1 0.000

MAJOR:        20.537  5.360  0.070  480.943
1 0.000
MINOR:        1414.474  0.000  0.000

UNIT: KN MET

YIELDING:  FORCE:  CAPACITY:  RATIO:  CRITERIA:  LOAD CASE:

LOCATION(MET):
MAJOR:       1.80E+02  3.10E+02  0.580  Cl. 13.5(a)
1 3.600
MINOR:       0.00E+00  1.46E+02  0.000  Cl. 13.5(a)
1 0.000

INTERMEDIATE:  Mp:  Se:  My:
MAJOR:        3.45E+02  0.00E+00  2.92E+03
MINOR:        1.62E+02  0.00E+00  1.46E+02

UNIT: KN MET

INTERACTION:  RATIO:  CRITERIA:  LOAD CASE:

LOCATION(MET):
FLEXURE AND AXIAL TENSION:
C/S STRENGTH:  0.580  Cl. 13.9.1  1  3.600
FLEXURE AND AXIAL COMPRESSION:
C/S STRENGTH:  0.801  Cl. 13.8.2  1  3.600
MEMBER STRENGTH:  0.896  Cl. 13.8.2  1  0.000
FLEX AND SHEAR:
BIAXIAL FLEX:
INTERMEDIATE:
FLEXURE AND AXIAL TENSION:
C/S STRENGTH:  2.92E+03  Mfz:  1.80E+02
3.10E+02
FLEXURE AND AXIAL COMPRESSION:
Cf:  Mfz:  Mfy:  Cr:  Mrz:  Mry:  b:
Cez:  Cey:  w 1z:  w 1y:  U1z:  U1y:
C/S STR:  9.00E+02  1.80E+02  0.00E+00  2.92E+03  3.10E+02  1.46E+02  0.85  1.76E+04  6.06E+03  0.60  0.60  1.00  1.00
V. India

V.IS801

V.IS 801-Beam with axial and major axis bending

Verification example for a cold-formed beam subject to axial compression and bending moment according to IS801.

Details

Verifies the calculations for an IS 60CU40x4 (Channel without lips) beam that is 2 m long and subject to axial compression and major axis bending moment. This example checks for compression, shear, bending and compression and bending interaction as per IS 801.

Material properties:

\[
\begin{align*}
E &= 2,074,000 \text{ kgf/cm}^2 = 203,400 \text{ MPa} \\
F_{y1} &= 350.0 \text{ MPa} = 3,569 \text{ kgf/cm}^2 \\
F_u &= 450 \text{ MPa} \\
G &= 795,000 \text{ kgf/cm}^2 = 77,963 \text{ MPa}
\end{align*}
\]

Design forces:

\[
\begin{align*}
P &= 0.5 \text{ kN} \\
M_z &= 0.748 \text{ kN m} \\
V_y &= 1.874 \text{ kN}
\end{align*}
\]

Verification

Section Dimension Checks

Check for flat width ratio:

\[
\begin{align*}
w &= b \cdot (r + t) = r \cdot (0.4 + 0.6) = 3 \text{ mm} \\
w/t &= 3 / 0.4 = 7.5 < 60
\end{align*}
\]
Hence, OK (ref. cl 5.2.3(a))
\[ h_w/t = (d - 2\times t_f - 2\times \text{root radius})/t = (6 - 2\times0.4 - 2\times0.6)/0.4 = 10 < 500 \]
Hence, OK (ref. cl 5.2.3(b))
Check for web height to thickness ratio:
\[ h/t = (6 - 2\times0.4)/0.4 = 13 < 150 \]
Hence, OK (ref. cl 5.2.4(a))
Check for limiting slenderness
\[ k_{Lyr} = 1.0 \frac{200 \text{ mm}}{1.263 \text{ mm}} = 158.3 < 200 \]
Hence, OK (ref. cl 6.3.3)

**Calculation of Allowable Compressive Stress**

For calculation of axially loaded member, “Q” is an important factor. The definition and method of calculation for value of Q is provided in Clause no. 6.6.1.1 (a). Channel section without lips is a combination of stiffened & unstiffened elements.

As per clause no 6.1.1.1 of IS801, the increase of steel strength happens due to cold work of forming.

Total corner area, \( A_{corner} = 2(45.2 \text{ mm}^2) = 91.2 \text{ mm}^2 = 0.912 \text{ cm}^2 \)

Total area of flanges, \( A_{flange} = 2\times b\times t = 2(0.4)(0.4) = 3.2 \text{ cm}^2 \)

\[ C = \frac{A_{corner}}{A_{flange}} = \frac{0.931}{3.2} = 0.285 \]

Effective depth, \( h_e = h = 4 \text{ cm} \)

Therefore, effective area, \( A_e = A = 4.91 \text{ cm}^2 \)

\[ B_e = 3.69 \left( \frac{F_u}{F_y} \right) \times 0.819 \left( \frac{F_u}{F_y} \right)^2 \times 1.79 = 1.6004 \]
\[ m = 0.192 \left( \frac{F_u}{F_y} \right) \times 0.068 = 0.1789 \]

Tensile yield point of corner, \( F_{yc} = \frac{F_u \times F_y}{r} \times m = 521.0 \text{ MPa} \)

Tensile yield point of flat portions, \( F_{yt} = F_y = 350.0 \text{ MPa} \)

Average yield point of cold-forming for tension/compression members, \( F_{ya(compression)} = (C \times F_{yc}) + (1 - C) \times F_{yc} = 381.8 \text{ MPa} \)

Average yield point of cold-forming for flexural members, \( F_{ya(bending)} = (C \times F_{yc}) + (1 - C) \times F_{yc} = 398.8 \text{ MPa} \)

As per cl. 6.2 of IS 801, compressive stress:
\[ F_c = 0.6 \times F_{ya} = 0.6 \times 381.8 = 229.1 \text{ MPa} \]
\[ Q_s = \frac{F_c}{0.6 \times F_{ya}} = \frac{229.1}{229.1} = 1 \]
Verification Examples

STAAD.Pro 3755 User Manual

\[ h = d - 2(r + t) = 4 \text{ cm} \]
\[ \frac{b}{t} = 13 < \frac{1435}{\sqrt{F_c}} = 29.69 \]
\[ Q_a = \frac{A_e}{A} = 1 \]
\[ Q = Q_a 	imes Q_d = 1 \]

Allowable compression stress, \( F_{a1} \), for members braced against twisting (ref. cl 6.6.1.1)

\[ C_e = \sqrt{\frac{2\pi E}{F_{ya}}} = 102.5 \]
\[ \frac{C_e}{\sqrt{Q}} = 102.5 \]

Slenderness ratio \( KL/r = 158.3 \)

\[ F_{a1} = \frac{12}{285} Q \times F_{ya} \times \frac{3(Q \times F_{ya})}{23a^2E} \left( \frac{KL}{r} \right)^2 = 41.06 \text{ MPa} \]

Maximum allowable compressive stress (Fa2) for flexural-torsional buckling (ref. cl 6.6.1.2 of IS 801)

\[ r_0 = \sqrt{(r_x^2) + (r_y^2) + (x_0^2)} = 3.732 \text{ cm} \]
\[ \sigma_x = \pi^2 \times \frac{E}{K \times L} = 274.8 \text{ MPa} \]
\[ \beta = 1 - \left( \frac{x_0}{r_x} \right)^2 = 0.5071 \]
\[ \sigma_t = \frac{1}{A \times r_o^2} \times \frac{(G \times J) + (\pi^2 \times E \times C_w)}{(K \times L_x)^2} = 324.2 \text{ MPa} \]
\[ \sigma_{TF0} = \frac{1}{2\beta} \left[ (\sigma_{ex} + \sigma_t) \times \sqrt{(\sigma_{ex} + \sigma_t)^2 - 4\beta \times \sigma_{ex} \times \sigma_t} \right] = 174.5 \text{ MPa} \]
\[ F_{Qy} = F_y \times Q = 381.8 \text{ MPa} \]
\[ F_{a2} = 0.522 \times \sigma_{TF0} = 91.09 \text{ MPa} \]

The allowable compressive stress, \( F_a \) is the minimum of \( F_{a1} \) and \( F_{a2} \):

\[ F_a = 41.06 \text{ MPa} \]

**Calculation of Allowable Bending Stress**

As per clause number 6.1, maximum allowable stress on extreme fiber is:

\[ F = 0.6 \times F_{ya(compression)} = 0.6 \times 381.8 \text{ MPa} = 229.1 \text{ MPa} \]

As the section is channel without lips, the flanges are unstiffened. So, as per clause 6.2 we need to check allowable compressive stress on the unstiffened element.

\[ \frac{w}{T} = 7.5 < \frac{530}{\sqrt{F_{ya}}} = \frac{530}{\sqrt{3893 \text{ kgf/cm}^2}} = 8.494 \]

Also, the yield strength of steel, \( F_y > 2,230 \text{ kgf/cm}^2 \) (= 227.5 MPa).
\[ F_c = 0.6 \times F_{ya(\text{compression})} = 229.1 \text{ MPa} \]

For the major axis bending, the allowable compressive stress, \( F_{bc} \), is the minimum of \( F \) and \( F_c \)

\[ F_{bc} = 229.1 \text{ MPa} \]

Similarly, for major axis bending, the allowable tensile stress, \( F_{bt} = 0.6 \times F_{ya(\text{compression})} = 229.1 \text{ MPa} \)

Calculate the allowable bending stress for laterally unbraced beams:

Unsupported length, \( L = 2 \text{ m} \) (the UNL parameter can be used for this).

\[ S_{xc} = Z_{xx} = 8.93 \text{ cm}^3 \]

\( C_b = 1.0 \) for a member under compression and bending.

\[
\frac{L}{d} \times \frac{S_{xc}}{I_{yc}} = 14,140 > 1.8n^2E \frac{C_b}{F_{ya(\text{compression})}} = 9,464
\]

\[ F_b = 0.6n^2E \times C_b \frac{d}{L} \times \frac{I_{yc}}{S_{xc}} = 85.18 \text{ MPa} \]

Allowable bending stress in the web:

\[ F_{bw1} = \frac{36,560,000}{h^2} = 216,330 \text{ kgf/cm}^2 = 21,215 \text{ MPa} \]

Per cl. 6.4.2, \( F_{bw} \) is the minimum of \( F_{bw1} \) and \( 0.6 \times F_{ya(bending)} = 239.2 \text{ MPa} \)

\[ F_{bw} = 239.2 \text{ MPa} \]

**Calculation of Allowable Shear Stress**

Per cl. 6.4:

Clear distance between flanges = \( h = d - 2t = 52 \text{ mm} \)

\[ F_{v1} = \frac{1275 \times \sqrt{F_y}}{h/t} = 577.1042 \text{ MPa} \quad \text{(cl. 6.4.1(a))} \]

\[ F_{v2} = \frac{585000}{h/t^2} = 3.395 \text{ MPa} \quad \text{(cl. 6.4.1(b))} \]

\[ h/t = 13 < \frac{4590}{\sqrt{F_y}} = 76.8 \]

Allowable shear stress, \( F_v \) is the minimum of \( F_{v1} \) or \( 0.4 \times F_y = 140.1 \text{ MPa} \)

\[ F_v = 140.1 \text{ MPa} \]

Allowable combined bending and shear stress:

\[ \frac{h}{t} < \frac{4590}{\sqrt{F_y}}, F_{vc} = F_{vcl} = 577.1 \text{ MPa} \]

**Actual Stresses**

Compression
f_a = P/A = 0.5 kN / 4.91 mm^2 = 1.018 MPa

Bending

f_b = M / Z_{xx} = 0.748 kN m / 8.93 cm^3 = 83.76 MPa

Bending in Web

Actual bending stress in the web is calculated by interpolation of bending stress diagram:

\[
f_{bw} = f_b \times \left(1 - \frac{t}{0.5 \times d}\right) = 72.59 \text{ MPa}
\]

Shear

\[
f_v = \frac{V}{N_c(t(d - 2t)^2)} = \frac{1.874(10)^3}{(1406 - 2 \times 4)} = 9.01 \text{ MPa}
\]

Stress Ratio

Compression

f_a / F_a = 1.018 / 41.06 = 0.024

Bending

for bending compression: f_b / F_{bc} = 83.76 / 229.1 = 0.366
for bending tension: f_b / F_{bt} = 83.76 / 229.1 = 0.366
for unbraced bending: f_b / F_b = 83.76 / 85.18 = 0.983
for web bending: f_{bw} / F_{bw} = 72.59 / 239.2 = 0.303

Shear

f_v / F_v = 9.01 / 140.1 = 0.064

Combined bending and shear (ref. cl 6.4.3 of IS 801):

\[
\sqrt{\left(\frac{f_{bw}}{F_{bw1}}\right)^2 + \left(\frac{f_v}{F_{vy1}}\right)^2} = \sqrt{\left(\frac{72.59}{2121.5}\right)^2 + \left(\frac{9.01}{577.1}\right)^2} = 0.016
\]

Interaction ratio for axial and bending

As Q = 1.0, F_{a0} can be calculated using cl. 6.6.1.1(b) with L = 0:

\[
F_{a0} = \frac{12}{23} Q \times F_{ya} \cdot \frac{3(Q \times F_{ya})^2}{23a^2 E} \left(\frac{K + L}{r}\right)^2 = 182.6 \text{ MPa}
\]

\[
\frac{f_a}{F_{a0}} + \frac{f_b}{F_{b1}} = \frac{1.018}{182.6} + \frac{83.76}{229.1} = 0.371
\]

Results

Table 523: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression stress ratio</td>
<td>0.024</td>
<td>0.024</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
### Result Type | Reference | STAAD.Pro | Difference | Comments
--- | --- | --- | --- | ---
Bending Z (compressive) stress ratio | 0.366 | 0.365 | negligible | 
Bending Z (tensile) stress ratio | 0.366 | 0.365 | negligible | 
Bending unbraced | 0.983 | 0.983 | none | 
Bending at web/flange junction stress ratio | 0.303 | 0.303 | none | 
Shear Y stress ratio | 0.064 | 0.064 | none | 
Compression + Bending interaction | 0.371 | 0.371 | none | 
Bending + Shear interaction | 0.016 | 0.016 | none | 

### STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 801\IS 801-Beam with axial and major axis bending.STD is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
  1 0 0 0; 2 2 0 0;
MEMBER INCIDENCES
  1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
  E 2.05e+08
  POISSON 0.3
  DENSITY 76.8195
  ALPHA 1.2e-05
  DAMP 0.03
  TYPE STEEL
  STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED INDIAN
  1 TABLE ST 60CU40X4
CONSTANTS
  MATERIAL STEEL ALL
  SUPPORTS
  1 FIXED
```
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -1.5
LOAD 2 LOADTYPE None TITLE LOAD CASE 3
MEMBER LOAD
1 CON GX -1
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
LOAD LIST 4
PARAMETER 1
CODE IS801
CWY 1 ALL
FU 450000 ALL
FYLD 350000 ALL
RATIO 1 ALL
TRACK 2 ALL
CHECK CODE ALL
STEEL TAKE OFF ALL
FINISH

**STAAD.Pro Output**

STAAD.Pro CODE CHECKING - (IS:801) v3.0

ALL UNITS ARE IN - METE KN (U.N.O.)

<table>
<thead>
<tr>
<th>MEMBER: 1</th>
<th>SECTION: 60CU40X4</th>
<th>LEN: 2.000</th>
<th>LOC: 0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>STATUS: PASS</td>
<td>RATIO: 0.983</td>
<td>REF: 6.3 LTB</td>
<td>LC: 4</td>
</tr>
</tbody>
</table>

DESIGN FORCES:

Fx:(C) 0.500  Fy: 1.874  Fz: 0.000
Mx: 0.000  My: 0.000  Mz: 0.748

SECTION PROPERTIES:

(Unit: CM)

| Ag: 4.91000 | Az: 3.20000 | Ay: 2.08000 |
| Cz: 1.38000 | Cy: 3.00000 |
| Iz: 26.80000 | Iy: 7.84000 | J: 0.25500 |
| Sz: 8.93000 | Sy: 2.99000 |
| Rz: 2.33629 | Ry: 1.26362 | Cw: 45.60001 |
## Verification Examples

### V.09 Steel Design

<table>
<thead>
<tr>
<th>MATERIAL INFO:</th>
<th>(Unit: MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fy: 350.025</td>
<td>Fu: 450.032</td>
</tr>
<tr>
<td>E: 203404.356</td>
<td>G: 77968.401</td>
</tr>
<tr>
<td>Fya(compression): 381.800</td>
<td>Fya(bending): 398.781</td>
</tr>
</tbody>
</table>

### DESIGN PROPERTIES:
- Member Length: 2.000
- Lz: 2.000
- Ly: 2.000
- Lb: 2.000

### DESIGN PARAMETERS:
- Kz: 1.000
- Ky: 1.000
- NSF: 1.000
- Cb: 0.000

### CRITICAL SLENDERNESS:
- Actual: 158.275
- Allowable: 200.000
- Ratio: 0.791

### CHECKS:

<table>
<thead>
<tr>
<th>Loc.</th>
<th>Demand</th>
<th>L/C</th>
<th>Actual</th>
<th>Allow</th>
<th>Ratio</th>
<th>Ref CL</th>
</tr>
</thead>
<tbody>
<tr>
<td>(MET)</td>
<td>(KN-MET)</td>
<td></td>
<td>(MPa)</td>
<td>(MPa)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tension</td>
<td>1.167</td>
<td>-0.50</td>
<td>4</td>
<td>1.018</td>
<td>229.080</td>
<td>0.004</td>
</tr>
<tr>
<td>Compression</td>
<td>0.000</td>
<td>0.50</td>
<td>4</td>
<td>1.018</td>
<td>41.835</td>
<td>0.024</td>
</tr>
<tr>
<td>BendZComp</td>
<td>0.000</td>
<td>0.75</td>
<td>4</td>
<td>83.688</td>
<td>229.080</td>
<td>0.365</td>
</tr>
<tr>
<td>BendZTens</td>
<td>0.000</td>
<td>0.75</td>
<td>4</td>
<td>83.688</td>
<td>229.080</td>
<td>0.365</td>
</tr>
<tr>
<td>BendUnbraced</td>
<td>0.000</td>
<td>0.75</td>
<td>4</td>
<td>83.688</td>
<td>85.160</td>
<td>0.983</td>
</tr>
<tr>
<td>BendYComp</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>239.268</td>
</tr>
<tr>
<td>BendYTens</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>229.080</td>
</tr>
<tr>
<td>Bend Web</td>
<td>0.000</td>
<td>0.75</td>
<td>4</td>
<td>72.529</td>
<td>239.268</td>
<td>0.303</td>
</tr>
<tr>
<td>Shear Z</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>140.010</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>1.87</td>
<td>4</td>
<td>9.009</td>
<td>140.010</td>
<td>0.064</td>
</tr>
<tr>
<td>Axial+Bend</td>
<td>0.000</td>
<td>-</td>
<td>4</td>
<td>-</td>
<td>-</td>
<td>0.371</td>
</tr>
<tr>
<td>6.7.2(a)2</td>
<td>Bend+Shear</td>
<td>0.000</td>
<td>-</td>
<td>4</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
V.IS 801-Column with axial and major axis bending

Verification example for a cold-formed column subject to axial compression and bending moment according to IS801.

Details

Verifies the calculations for an IS 60CS40x4 (channel with lips) column that is 2 m long and subject to axial compression and major axis bending moment. This example checks for compression, shear, bending, and compression and bending interaction as per IS 801.

Material properties:

\[ E = 2,074,000 \text{ kgf/cm}^2 = 203,400 \text{ MPa} \]
\[ F_y = 3,569 \text{ kgf/cm}^2 = 353.0 \text{ MPa} \]
\[ F_u = 4,588 \text{ kgf/cm}^2 = 450 \text{ MPa} \]
\[ G = 795,000 \text{ kgf/cm}^2 = 77,963 \text{ MPa} \]

Design forces:

\[ P = 10 \text{ kN} \]
\[ M_z = 4.0 \text{ kN·m} \]
\[ V_y = 2 \text{ kN} \]

Verification

Section Dimension Checks

Flanges (ref. cl 5.2.3):

\[ w = 4 \times 2 \times 0.4 \times 2 \times 0.6 = 2 \]
\[ w/t = 2 / 0.4 = 5 < 60, \text{ Hence OK} \]

Web (ref. cl 5.2.4):

\[ W/t = (6 \times 2 \times 0.4 \times 2 \times 0.6) / 0.4 = 10 < 500 \text{ and} \]
\[ h/t = (6 \times 2 \times 0.4) / 0.4 = 13 < 150, \text{ Hence OK} \]

Check for limiting slenderness (ref. cl 6.3.3):

\[ K \times L_y/r_y = 1.0 \times 200 \text{ mm} / 1.453 \text{ mm} = 137.6 < 200 \]

Hence, OK
**Allowable Compression Strength**

Maximum allowable compressive stress for flexural buckling $Y$ (ref. cl 6.6.1.1 of IS 801)

Design stress, $F = 0.6 \times f_y = 2,160 \text{ kgf/cm}^2 = 211.8 \text{ MPa}$

Effective width of flange, $B_{eff}$

\[ W = 4 - 2 \times 0.4 - 2 \times 0.6 = 2 \text{ cm} \]
\[ \frac{w}{t} = 5 < \frac{1.435}{\sqrt{F}} = 30.87 \]
\[ B_{eff} = w = 2 \text{ cm} \]

Effective width of web, $B_{eff}$

\[ W/t < 30.87 \]
\[ B_{eff} = w = 10 \text{ cm} \]
\[ A_{eff} = 5.8 \text{ cm}^2 \]

Factor, $Q = A_{eff} / A = 1.0 = C_e = \frac{2}{\pi} \sqrt{\frac{E}{F_y}} = 106.6$

Slenderness limit $= \frac{C_e}{Q} = 106.6 < 137.6$

\[ F_{a1} = \frac{10,680,000}{(kL/r)^2} = 564.1 \text{ kgf/cm}^2 \]

Maximum allowable compressive stress for flexural torsional buckling (ref. cl 6.6.1.2 of IS 801)

Polar radius of gyration, $r_0 = \sqrt{r_x^2 + r_y^2 + x_0^2} = 5.181 \text{ cm}$

\[ \beta = 1 - \left(\frac{x_0}{r_0}\right)^2 = 0.259 \]
\[ \sigma_t = 2,036 \text{ kgf/cm}^2 \]
\[ \sigma_{ex} = 2,480 \text{ kgf/cm}^2 \]

\[ \sigma_{TP0} = \frac{1}{2F_y} \left[ (\sigma_{ex} + \sigma_t) \cdot \sqrt{(\sigma_{ex} + \sigma_t)^2 - 4 \beta \times \sigma_{ex} \times \sigma_t} \right] = 1,201 \text{ kgf/cm}^2 = 117.8 \text{ MPa} \]

1,201 kgf/cm$^2$ $< 0.5 \times F = 1,800 \text{ kgf/cm}^2$

\[ F_{a2} = 0.522 \times \sigma_{TP0} = 626.9 \text{ kgf/cm}^2 \]

The allowable compressive stress, $F_a$ is the minimum of $F_{a1}$ and $F_{a2}$.

\[ F_a = 546.1 \text{ kgf/cm}^2 = 55.32 \text{ MPa} \]

**Allowable Bending Strength**

$C = 1.0$ for member under combined axial and bending.

\[ F_b = \frac{2}{5} F_y \cdot \frac{F_y}{54\pi^2 E} \cdot C_b \left( \frac{L^2 S_{xc}}{d \times t_{yc}} \right) = 1,412 \text{ kgf/cm}^2 = 138.5 \text{ MPa} \]  

(ref. cl 6.3)

Maximum allowable compressive stress:
F_b1 = 0.6 \times F_y = 2,160 \text{ kgf/cm}^2 = 211.8 \text{ MPa}

**Allowable Shear Strength**

\[
\frac{h}{t} = 13 < \frac{4.590}{\sqrt{F_y}} = 76.5
\]

\[
F_{v1} = \frac{1.275 \cdot \sqrt{F_y}}{(h/t)} = 5,884 \text{ kgf} \mid \text{cm}^2
\]

\[
F_{v2} = 0.4F_y = 1,440 \text{ kgf} \mid \text{cm}^2
\]

The allowable shear stress, \(F_v\), is the minimum of \(F_{v1}\) and \(F_{v2}\):

\[
F_v = 1,440 \text{ kgf/cm}^2 = 141.3 \text{ MPa}
\]

**Axial and Bending Interaction**

\[
f_a = \frac{P}{A} = 10 \text{ kN} / 5.8 \text{ cm}^2 = 1,019.7 \text{ kgf} / 5.8 \text{ cm}^2 = 175.8 \text{ kgf/cm}^2
\]

\[
f_{b1} = \frac{M}{Z} = 4.0 \text{ kN} \cdot \text{m} / 9.4 \text{ cm}^3 = 40,789 \text{ kgf} \cdot \text{cm} / 9.4 \text{ cm}^3 = 4,339 \text{ kgf/cm}^2
\]

\[
C_{mx} = 0.6
\]

\[
F'_e = 1,294 \text{ kgf/cm}^2
\]

\[
\frac{f_a}{F_{a1}} + \frac{f_{b1} C_{m}}{F_{b1}} = \frac{175.8}{564.1} + \frac{4,390(0.6)}{2,160} = 0.312 + 1.395 = 1.705
\]

Hence, not OK

\[
F_{a0} = 1,879 \text{ kgf/cm}^3
\]

\[
\frac{f_a}{F_{a0}} + \frac{f_{b1}}{F_{b1}} = \frac{175.8}{1,879} + \frac{439}{2,160} = 0.094 + 2.009 = 2.103
\]

**Results**

**Table 524: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable compressive stress (MPa)</td>
<td>55.32</td>
<td>55.371</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (MPa)</td>
<td>211.8</td>
<td>211.854</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable unbraced bending stress (MPa)</td>
<td>138.5</td>
<td>138.583</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress along Y (MPa)</td>
<td>141.3</td>
<td>141.236</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>
### STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 801\IS 801-Column with axial and major axis bending.STD is typically installed with the program.

**STAAD SPACE**

**START JOB INFORMATION**

**ENGINEER DATE** 18-Mar-19

**END JOB INFORMATION**

**INPUT WIDTH** 79

**UNIT** METER KN

**JOINT COORDINATES**

1 0 0 0; 2 0 2 0;

**MEMBER INCIDENCES**

1 1 2;

**DEFINE MATERIAL START**

**ISOTROPIC STEEL**

E 2.05e+08

POISSON 0.3

DENSITY 76.8195

ALPHA 1.2e-05

DAMP 0.03

**TYPE STEEL**

**STRENGTH RY 1.5 RT 1.2**

**END DEFINE MATERIAL**

**MEMBER PROPERTY** COLDFORMED INDIAN

1 TABLE ST 60CS40X4

**CONSTANTS**

**MATERIAL STEEL ALL**

**SUPPORTS**

1 FIXED

**LOAD 1 LOADTYPE None** TITLE LOAD CASE 1

**JOINT LOAD**

2 FX 2 FY -10

**PERFORM ANALYSIS**

**PRINT** STATICS CHECK

**PARAMETER 1**

**CODE IS801**

**TRACK 2 ALL**

**CHECK CODE ALL**

**PRINT MEMBER PROPERTIES ALL**

**FINISH**

### STAAD.Pro Output

**STAAD.Pro CODE CHECKING - ( IS:801 ) v3.0**

*ALL UNITS ARE IN - METE KN (U.N.O.)*

**MEMBER: 1 SECTION: 60CS40X4 LEN: 2.000 LOC:**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial and Bending interaction</td>
<td>2.103</td>
<td>2.102</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>
Verification Examples
V.09 Steel Design

| STATUS: | RATIO: 3.071 | REF: 6.3 LTB | LC: 1 |

Design Forces:
Fx:(C) 10.000  Fy: 2.000  Fz: 0.000
Mx: 0.000  My: 0.000  Mz: 4.000

Section Properties:
Ag: 5.82000  Az: 3.20000  Ay: 3.36000
Cz: 1.75000  Cy: 3.00000
Iz: 28.20000  Iy: 12.30000  J: 0.29600
Sz: 9.40000  Sy: 5.46000
Rz: 2.20122  Ry: 1.45375  Cw: 162.00003

Material Info:
Fy: 353.090  Fu: 449.993  E: 203404.356  G: 77968.401
Fya(compression): 353.090  Fya(bending): 353.090

Design Properties:
Member Length: 2.000  Lz: 2.000  Ly: 2.000  Lb: 2.000

Design Parameters:
Kz: 1.000  Ky: 1.000  NSF: 1.000  Cb: 0.000

Critical Slenderness:
Actual: 137.575  Allowable: 200.000  Ratio: 0.688

Checks:
<table>
<thead>
<tr>
<th>Loc.</th>
<th>Demand</th>
<th>L/C</th>
<th>Actual</th>
<th>Allow</th>
<th>Ratio</th>
<th>Ref CL</th>
</tr>
</thead>
<tbody>
<tr>
<td>(MET)</td>
<td>(KN-MET)</td>
<td>(MPa)</td>
<td>(MPa)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Verification Examples

**V.09 Steel Design**

<p>| | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Tension</strong></td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>211.854</td>
<td>-</td>
<td>6.1</td>
</tr>
<tr>
<td><strong>Compression</strong></td>
<td>0.000</td>
<td>10.00</td>
<td>1</td>
<td>17.183</td>
<td>55.371</td>
<td>0.310</td>
<td>6.6.1.1</td>
</tr>
<tr>
<td><strong>BendZComp</strong></td>
<td>0.000</td>
<td>4.00</td>
<td>1</td>
<td>425.562</td>
<td>211.854</td>
<td>2.009</td>
<td>6.3</td>
</tr>
<tr>
<td><strong>BendZTens</strong></td>
<td>0.000</td>
<td>4.00</td>
<td>1</td>
<td>425.562</td>
<td>211.854</td>
<td>2.009</td>
<td>6.3</td>
</tr>
<tr>
<td><strong>BendUnbraced</strong></td>
<td>0.000</td>
<td>4.00</td>
<td>1</td>
<td>425.562</td>
<td>138.583</td>
<td>3.071</td>
<td>6.3 LTB</td>
</tr>
<tr>
<td><strong>BendYComp</strong></td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>211.854</td>
<td>-</td>
<td>6.3</td>
</tr>
<tr>
<td><strong>BendYTens</strong></td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>211.854</td>
<td>-</td>
<td>6.3</td>
</tr>
<tr>
<td><strong>Bend Web</strong></td>
<td>0.000</td>
<td>4.00</td>
<td>1</td>
<td>368.820</td>
<td>211.854</td>
<td>1.741</td>
<td>6.4.2</td>
</tr>
<tr>
<td><strong>Shear Z</strong></td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>141.236</td>
<td>-</td>
<td>6.3</td>
</tr>
<tr>
<td><strong>Shear Y</strong></td>
<td>0.000</td>
<td>2.00</td>
<td>1</td>
<td>5.953</td>
<td>141.236</td>
<td>0.042</td>
<td>6.4.1</td>
</tr>
<tr>
<td><strong>Axial+Bend</strong></td>
<td>0.000</td>
<td>-</td>
<td>1</td>
<td>-</td>
<td>2.102</td>
<td>-</td>
<td>2.102</td>
</tr>
<tr>
<td><strong>Bend+Shear</strong></td>
<td>0.000</td>
<td>-</td>
<td>1</td>
<td>-</td>
<td>0.020</td>
<td>-</td>
<td>6.4.3</td>
</tr>
</tbody>
</table>

Effective Section Properties:(cm)

| Ae: 5.820 | SzTop: 9.400 | SzBot: 9.400 | SyLeft: 7.029 | SyRight: 5.467 |

Intermediate Results: Cb = 1.000

---

NOTE: Torsion and deflections have not been considered in the design.

### V. Japan

#### V.AIJ 2002

**V.AIJ 2002 Check for MBG parameter**

Verify the effect of the MBG parameter of the AIJ 2002 code on a doubly symmetric I shaped beam of span 6m, that is subjected to UDL of -20 KN/m on its span.

**Details**

Japanese H Shape Doubly Symmetric I Section H400X200X12X22 used in the following frame, E = 205,000 MPa, $F_y = 235$ MPa
where $w = -20 \text{kN/m}$ and $L = 4 \text{ m}$

Equation SSB-1.10 of JSME is used to evaluate Young’s modulus, $E$, for equation 5.8 ($\text{YNG ~1}$).

d = 400 mm, $b_f = 200 \text{ mm}$, $t_w = 12 \text{ mm}$, $t_f = 22 \text{ mm}$

**Verification**

Maximum Moment, $MZ_1 = 43.714 \text{ kN}\cdot\text{m}$

Minimum Moment, $MZ_2 = 43.714 \text{ kN}\cdot\text{m}$

Span Moment = $46.286 \text{ kN}\cdot\text{m}$

Unbraced length of compression flange, $l_b = 6,000 \text{ mm}$

**Note:** Only the flanges will be considered (i.e., the web is ignored for calculating the section modulus). ($\text{MBG ~1}$)

Area of compression flange = $A_f = b_f \times t_f = 4,400 \text{ mm}^2$

$$C_g = \frac{b_f \times t_f \times \left( d - \frac{t_f}{2} \right) + b_f \times t_f \times \frac{t_f}{2}}{(b_f \times t_f) + (b_f \times t_f)} = 200 \text{ mm}$$

$$I_{zz} = 2 \times \left( b_f \times \frac{t_f^3}{12} \right) + 2 \times b_f \times t_f \times \left( c_g \cdot \frac{t_f}{2} \right)^2 = 3.147 \times (10)^8 \text{ mm}^4$$

Elastic Section Modulus about Z axis on compression side $Z_{zc} = I_{zz}/c_g = 1.573 \times 10^6 \text{ mm}^3$

$I_{yc} = t_f \times b_f^3/12 = 1.467 \times 107 \text{ mm}^4$

$A_C = t_f \times b_f = 4,400 \text{ mm}^2$

$r_i = \sqrt{I_{yc}/A_C} = 57.735 \text{ mm}$
Bending Moment Factor $C = 1.75 - 1.05 \times (-MZ2/MZ1) + 0.3 \times (-MZ2/MZ1)^2 = 3.198$, but not more than 2.3

As Span moment $> MZ1$, $C$ will be 1

Allowable slenderness ratio $\Lambda = \sqrt{\frac{n^2 \times E}{0.6 \times F_{YLD}}} = 119.789$

$$f_{b1} = 1.5 \times \left[ 1 \times 0.4 \times \left( \frac{l_b}{C \times A} \right)^2 \right] \times \frac{f_t}{\gamma} = 109.501 \text{ N/mm}^2$$

As per Eq. SSB-1.10 for $YNG = 1$:

$$f_{b2} = 1.5 \times \frac{0.433 \times E}{l_b \times d} = 162.736 \text{ N/mm}^2$$

$$f_{bz1} = \max (f_{b1}, f_{b2}) = 162.736 \text{ N/mm}^2$$

Allowable tensile bending stress, $f_{bzt} = f_{t} = 156.667 \text{ N/mm}^2$

Actual tensile bending stress about Z axis = $46.286/Z_{ZC} = 29.425 \text{ N/mm}^2$

Utilization ratio = Actual bending stress/Allowable bending stress = $29.425/156.667 = 0.1878$

Allowable compressive bending stress, $f_{bzc} = \min (f_{bz1}, f_{t}) = 156.667 \text{ N/mm}^2$

Actual compressive bending stress about Z axis = $46.286/Z_{ZC} = 29.425 \text{ N/mm}^2$

Utilization ratio = Actual bending stress/Allowable bending stress = $29.425/156.667 = 0.1878$

Results

Table 525: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actual bending tensile stress (N/mm$^2$)</td>
<td>29.425</td>
<td>29.42</td>
<td></td>
<td>Negligible</td>
</tr>
<tr>
<td>Allowable bending tensile stress (N/mm$^2$)</td>
<td>156.667</td>
<td>156.67</td>
<td></td>
<td>Negligible</td>
</tr>
<tr>
<td>Ratio</td>
<td>0.1878</td>
<td>0.188</td>
<td></td>
<td>Negligible</td>
</tr>
<tr>
<td>C</td>
<td>1</td>
<td>1</td>
<td></td>
<td>None</td>
</tr>
<tr>
<td>Actual bending compressive stress (N/mm$^2$)</td>
<td>29.425</td>
<td>29.42</td>
<td></td>
<td>Negligible</td>
</tr>
<tr>
<td>Allowable bending compressive stress (N/mm$^2$)</td>
<td>156.667</td>
<td>156.67</td>
<td></td>
<td>Negligible</td>
</tr>
</tbody>
</table>
### STAAD.Pro Input File

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\AIJ\2002\AIJ 2002 Check for MBG parameter.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 28-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 6 4 0; 4 6 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT METER KN
ANGLE
ANGLE
0.2 0.2 0.018 0.0394179 0.0024 0.0024
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
1 TO 3 TABLE ST H400X200X12X22
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -20
PERFORM ANALYSIS
DEFINE ENVELOPE
1 ENVELOPE 1 TYPE STRENGTH
END DEFINE ENVELOPE
LOAD LIST ENV 1
PARAMETER 1
CODE JAPANESE 2002
TRACK 2 MEMB 2
MAIN 1 MEMB 2
MISES 1 MEMB 2
YNG 1 MEMB 2
```
STAAD.Pro Output

STAAD.Pro CODE CHECKING - (AIJ-2002) V2.1

*******************************
ALL UNITS ARE IN - METE KN (U.N.O.)

Member No: 2  Profile: ST H400X200X12X22 (JAPANESE SECTIONS)
Ratio: 0.231 (PASS) Reference: Eq.5.16  Loadcase: 1

Location: 0.000  Criteria: Stress  Load Case: 1

Px:(C) 15.749  Vy: 60.000  Vz: 0.000
Tx: 0.000  My: 0.000  Mz: 43.714

SECTION PROPERTIES AT DISTANCE: 0.000 MM (UNIT: MM)

Ax: 1.3220E+04  Az: 5.86667E+03  Ay: 4.80000E+03
Iz: 3.64000E+08  Iy: 2.94000E+07  J: 1.64000E+06
Zz: 1.82000E+06  Zy: 2.94000E+05  Zx: 7.45455E+04
iZ: 1.65934E+02  iY: 4.71583E+01  Iw: 1.05020E+12

MATERIAL PROPERTIES (Unit: N/mm²)

Fy: 235.000  E: 204999.987  G: 79000.000

DESIGN PROPERTIES

Member Length: 6.000  Lz: 6.000  Ly: 6.000  UNL: 6.000

DESIGN PARAMETERS

Kz: 1.000  Ky: 1.000  NSF: 1.000  Cb: 1.000

CRITICAL SLENDERNESS (Tension)

Actual: 0.000  Allowable: 1.000  Ratio: 0.000
### Verification Examples

**V.09 Steel Design**

---

#### CHECKS

<table>
<thead>
<tr>
<th>Loc.</th>
<th>Demand</th>
<th>L/C</th>
<th>Typ</th>
<th>Actual</th>
<th>Allow</th>
<th>Ratio</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td>(MET)</td>
<td>(KN-MET)</td>
<td>(N/mm²)</td>
<td>(N/mm²)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Checks</th>
<th>Loc.</th>
<th>Demand</th>
<th>L/C</th>
<th>Typ</th>
<th>Actual</th>
<th>Allow</th>
<th>Ratio</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tension</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>156.67</td>
<td>-</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Compression</td>
<td>0.000</td>
<td>15.75</td>
<td>1</td>
<td>P</td>
<td>29.42</td>
<td>156.67</td>
<td>0.188</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Bending Z (T)</td>
<td>3.000</td>
<td>-46.29</td>
<td>1</td>
<td>P</td>
<td>29.42</td>
<td>156.67</td>
<td>0.188</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Bending Z (C)</td>
<td>3.000</td>
<td>-46.29</td>
<td>1</td>
<td>P</td>
<td>29.42</td>
<td>156.67</td>
<td>0.188</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Bending Y (T)</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>156.67</td>
<td>-</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Bending Y (C)</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>156.67</td>
<td>-</td>
<td>Eq.5.1</td>
</tr>
<tr>
<td>Shear Z</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>90.45</td>
<td>-</td>
<td>Eq.5.2</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>60.00</td>
<td>1</td>
<td>P</td>
<td>12.50</td>
<td>90.45</td>
<td>0.138</td>
<td>Eq.5.2</td>
</tr>
<tr>
<td>Comp+ Bend C</td>
<td>3.000</td>
<td>-</td>
<td>1</td>
<td>P</td>
<td>-</td>
<td>-</td>
<td>0.208</td>
<td>Eq.6.1</td>
</tr>
<tr>
<td>Comp+ Bend T</td>
<td>3.000</td>
<td>-</td>
<td>1</td>
<td>P</td>
<td>-</td>
<td>-</td>
<td>0.180</td>
<td>Eq.6.2</td>
</tr>
<tr>
<td>Ten + Bend T</td>
<td>3.000</td>
<td>-</td>
<td>1</td>
<td>P</td>
<td>-</td>
<td>-</td>
<td>0.188</td>
<td>Eq.6.3</td>
</tr>
<tr>
<td>Ten + Bend C</td>
<td>3.000</td>
<td>-</td>
<td>1</td>
<td>P</td>
<td>-</td>
<td>-</td>
<td>0.188</td>
<td>Eq.6.4</td>
</tr>
<tr>
<td>Von-Mises</td>
<td>0.000</td>
<td>-</td>
<td>1</td>
<td>P</td>
<td>36.17</td>
<td>156.67</td>
<td>0.231</td>
<td>Eq.5.16</td>
</tr>
</tbody>
</table>

#### INTERMEDIATE RESULTS (Bending)

**C:** 1.000

#### INTERMEDIATE RESULTS (Von-Mises)

- **Sigmax:** 28.972 N/mm²
- **Tou:** 12.500 N/mm²
- **fm:** 36.168 N/mm²

---

**V.AIJ 2002 Check for MISES parameter**

Verify the von Mises stress calculations per the AIJ 2002 code using a cantilever beam.
Details

Japanese Angle section L250X250X35, $E = 205,000$ MPa, $F_y = 200$ MPa

The member is a 5 m cantilever with the following applied loads at the end:

- $F_x = 10$ kN
- $V_z = 5$ kN
- $V_y = 5$ kN
- $T = 5$ kN-m

Verification

The maximum of the left-hand side of the von Mises stress equation apparently occurs at the fixed support of the beam. Section forces at the fixed end are calculated as below.

- $F_X = -10$ KN
- $F_Y = -5$ KN
- $F_Z = -5$ KN
- $M_X = -5$ KN-m
- $M_Y = 25$ KN-m
- $M_Z = -25$ KN-m

The standard AIJ calculation is used ($MISES ~1$). From these section forces, $\sigma_x$ and $\tau_{xy}$ at the section of the fixed end are calculated:

$$\sigma_x = \sqrt{\frac{F_x}{A_x} + \frac{M_y}{Z_y} + \frac{M_z}{Z_z}}$$

$$= \sqrt{\frac{-10,000}{16,260} + \frac{25,000,000}{838,660} + \frac{-25,000,000}{355,900}}$$

$$= 0.615 + 29.809 + 70.244 = 100.668 \text{ N/mm}^2$$

$$\tau_{xy} = \sqrt{\frac{M_x}{A_x} + \frac{F_y}{A_y}^2 + \frac{F_z}{A_z}^2}$$

$$= \sqrt{\frac{5,000,000}{189,700} + \frac{5,000}{5,833}^2 + \frac{5,000}{5,833}^2}$$

$$= 27.57 \text{ N/mm}^2$$

From $\sigma_x$ and $\tau_{xy}$, $f_m$ is calculated:

$$f_m = \sqrt{\sigma_x^2 + 3\tau_{xy}^2} = 111.42 \text{ N/mm}^2$$

For permanent loading, $k = 1$ and $f_t = f_y / 1.5 = 133.33 \text{ N/mm}^2$

Therefore, the ratio $= f_m / k \times f_t = 111.419 / (1 \times 133.33) = 0.836$
Results

Table 526: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>( f_m ) (N/mm(^2))</td>
<td>111.42</td>
<td>111.42</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>( f_t ) (N/mm(^2))</td>
<td>133.33</td>
<td>133.33</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>( \sigma_x ) (N/mm(^2))</td>
<td>100.668</td>
<td>100.669</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \tau_{xy} ) (N/mm(^2))</td>
<td>27.569</td>
<td>27.57</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Ratio</td>
<td>0.836</td>
<td>0.836</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\V.09 Steel Design\AIJ\2002\AIJ 2002 Check for MISES parameter.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Apr-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
1 TABLE ST L250X250X35
*1 TABLE ST H100X50X5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FX 10 FY 5 FZ 5 MX 5
PERFORM ANALYSIS PRINT STATICS LOAD
PRINT MEMBER PROPERTIES ALL
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE JAPANESE 2002
*TMP 1 ALL
STAAD.Pro CODE CHECKING - (AIJ-2002) V2.1

***************

ALL UNITS ARE IN - METE  KN   (U.N.O.)

---

<table>
<thead>
<tr>
<th>Member No:</th>
<th>1</th>
<th>Profile: ST L250X250X35 (JAPANESE SECTIONS)</th>
<th>Ratio: 0.836 (PASS)</th>
<th>Reference: Eq.5.16</th>
<th>Loadcase: 1</th>
</tr>
</thead>
</table>

---

<table>
<thead>
<tr>
<th>Location: 0.000</th>
<th>Criteria: Stress</th>
<th>Load Case: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Px:(T) -10.000</td>
<td>Vy: -5.000</td>
<td>Vz: -5.000</td>
</tr>
<tr>
<td>Tx: -5.000</td>
<td>My: 25.000</td>
<td>Mz: -25.000</td>
</tr>
</tbody>
</table>

---

SECTION PROPERTIES AT DISTANCE: 0.000 MM (UNIT: MM)

<table>
<thead>
<tr>
<th>Ax: 1.62600E+04</th>
<th>Az: 5.83333E+03</th>
<th>Ay: 5.83333E+03</th>
</tr>
</thead>
<tbody>
<tr>
<td>Iz: 3.79328E+07</td>
<td>Iy: 1.48256E+08</td>
<td>J: 6.63950E+06</td>
</tr>
<tr>
<td>Zz: 3.55901E+05</td>
<td>Zy: 8.38661E+05</td>
<td>Zx: 1.89700E+05</td>
</tr>
<tr>
<td>iz: 4.83000E+01</td>
<td>iy: 9.54873E+01</td>
<td>Iw: 2.99365E+10</td>
</tr>
</tbody>
</table>

---

MATERIAL PROPERTIES (Unit: N/mm2)

| Fy: 200.000 | E: 205000.000 | G: 79000.000 |

---

DESIGN PROPERTIES

<table>
<thead>
<tr>
<th>Member Length: 5.000</th>
<th>Lz: 5.000</th>
<th>Ly: 5.000</th>
<th>UNL: 5.000</th>
</tr>
</thead>
</table>

---

DESIGN PARAMETERS

<table>
<thead>
<tr>
<th>Kz: 1.000</th>
<th>Ky: 1.000</th>
<th>NSF: 1.000</th>
<th>Cb: 0.000</th>
</tr>
</thead>
</table>
## CRITICAL SLENDERNESS (Tension)

<table>
<thead>
<tr>
<th>Actual</th>
<th>Allowable</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>103.52</td>
<td>400.00</td>
<td>0.259</td>
</tr>
</tbody>
</table>

### CHECKS

<table>
<thead>
<tr>
<th>Env</th>
<th>Stresses</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Tension
- Demand: -10.00
- Actual: 0.62
- Allow: 133.33
- Ratio: 0.005
- Ref: Eq.5.1

#### Compression
- Demand: -
- Actual: -
- Allow: 77.53
- Ref: Eq.5.3

#### Bending Z (T)
- Demand: -25.00
- Actual: 70.24
- Allow: 133.33
- Ratio: 0.527
- Ref: Eq.5.1

#### Bending Z (C)
- Demand: -25.00
- Actual: 70.24
- Allow: 133.33
- Ratio: 0.527
- Ref: Eq.5.1

#### Bending Y (T)
- Demand: 25.00
- Actual: 29.81
- Allow: 133.33
- Ratio: 0.224
- Ref: Eq.5.1

#### Bending Y (C)
- Demand: 25.00
- Actual: 29.81
- Allow: 133.33
- Ratio: 0.224
- Ref: Eq.5.1

#### Shear Z
- Demand: -5.00
- Actual: 0.86
- Allow: 76.98
- Ratio: 0.011
- Ref: Eq.5.2

#### Shear Y
- Demand: -5.00
- Actual: 0.86
- Allow: 76.98
- Ratio: 0.011
- Ref: Eq.5.2

#### Comp+ Bend C
- Demand: -
- Actual: -
- Allow: -
- Ratio: 0.750
- Ref: Eq.6.1

#### Comp+ Bend T
- Demand: -
- Actual: -
- Allow: -
- Ratio: 0.750
- Ref: Eq.6.2

#### Ten + Bend T
- Demand: -
- Actual: -
- Allow: -
- Ratio: 0.755
- Ref: Eq.6.3

#### Ten + Bend C
- Demand: -
- Actual: -
- Allow: -
- Ratio: 0.746
- Ref: Eq.6.4

#### Von-Mises
- Demand: -
- Actual: 111.42
- Allow: 133.33
- Ratio: 0.836
- Ref: Eq.5.16

### INTERMEDIATE RESULTS (Bending)

- C: 0.000

### INTERMEDIATE RESULTS (Von-Mises)

- Sigmax: 100.669 N/mm²
- Tou: 27.570 N/mm²
- fm: 111.420 N/mm²

### Related Links

- [D9.C.10 Von Mises Stresses Check](on page 2001)
V. New Zealand

V. NZS 3404 1997

V. NZS3404 1997-UB Section
Verify the Design Capacity of I Section as per NZS3404 1997.

Details
Verify the section bending capacity of an UB530X92.4. The member is used in a 15 m simply supported span.

Validation

Section Classification
Evaluate the slenderness the beam flanges:

\[ \lambda_{ef} = \left( \frac{B}{2t_f} \right) \times \sqrt{\left( \frac{f_y}{250} \right)} = \frac{198.8}{(2 \times 15.60)} \times \sqrt{300/250} = 6.98 < 8 \]

Flange is compact.

Evaluate the slenderness of the beam web:

\[ \lambda_{ew} = \left( \frac{d}{t_w} \right) \times \sqrt{\left( \frac{f_y}{250} \right)} = \frac{501.80}{10.20} \times \sqrt{300/250} = 53.89 < 89 \]

Web is compact.

Bending Capacity

Section Bending Capacity About Strong Axis
Effective section modulus, \( Z_{ez} = 2.37(10)^6 \text{ mm}^3 \)

The nominal section capacity in bending about the Z axis, \( M_{sz} = \phi f_y \times Z_{ez} \)

\[ M_{sz} = 0.9 \times 300 \times 2.37 = 639.9 \text{ kN m} \]

Section Bending Capacity About Weak Axis
Effective section modulus, \( Z_{ey} = 341.6(10)^3 \text{ mm}^3 \)

The nominal section capacity in bending about the Y axis, \( M_{sy} = \phi f_y \times Z_{ey} \)

\[ M_{sy} = 0.9 \times 300 \times 341.6 \times (10)^3 = 92.24 \text{ kN m} \]

Member Bending Capacity
End restraint arrangement = FF
A twist restraint factor, \( K_t(SKT) = 1.00 \)
Minor axis rotation restraints = Both
Lateral rotation restraint factor, \( K_r(SKR) = 1 \)
Load Height factor, \( K_l(LHT) = 1.00 \) (Table 5.6.3(2) of NZS3404:1997)
Effective length = 1×1×1×15,000 = 15,000 mm

Design Bending Moment, \( M_m = 160.8 \text{ kN} \cdot \text{m} \)

Quarter Point Moment of segment, \( M_2 = -72.443 \text{ kN} \cdot \text{m} \)

Mid-Point Moment of segment, \( M_3 = -2.267 \text{ kN} \cdot \text{m} \)

Quarter Point Moment of segment, \( M_4 = 72.499 \text{ kN} \cdot \text{m} \)

\[ \alpha_m = \frac{1.7M_m}{\sqrt{M_1^2 + M_2^2 + M_3^2}} \leq 2.5 \]

\[ \alpha_m = \frac{1.7 \times 160.8}{\sqrt{(-72.443)^2 + (-2.267)^2 + (72.499)^2}} = 2.66 \]

Therefore, \( \alpha_m = 2.5 \)

Reference Buckling Moment, \( M_o \)

\[ M_o = \sqrt{\frac{n^2EI_y}{L_e^2}} \left[ GJ + \left( \frac{n^2EI_y}{L_e^2} \right) \right] = 127.0 \text{ kN} \cdot \text{m} \]

\[ \alpha_s = 0.6 \left[ \left( \frac{M_{sx}}{M_{oa}} \right)^2 + 3 \right] \left( \frac{M_{sx}}{M_{oa}} \right) = 0.157 \]

\[ M_{bz} = \alpha_m \alpha_s M_{sx} \leq M_{sx} = 2.5 \times 0.157 \times 71 = 279.1 \text{ kN} \cdot \text{m} \]

\[ \phi_{M_{bz}} = 251.2 \text{ kN} \cdot \text{m} \]

**Check for Shear**

Shear Area of the section, \( A_y = d \times t_w = 533.0 \times 10.2 = 5,436.6 \text{ mm}^2 \)

Section Shear Capacity (Along Y axis), \( V_y = 0.6 \times f_y \times A_y = 0.6 \times 320 \times 5,436.6 = 1,044 \text{ kN} \)

\( ZV_y = 0.9 \times 1,044 = 939.4 \text{ kN} \)

Shear Area of the section, \( A_Z = 2 \times b_f \times t_f = 2 \times 209 \times 15.6 = 6,520.8 \text{ mm}^2 \)

Section Shear Capacity (Along Z axis), \( V_z = 0.6 \times f_y \times A_Z = 0.6 \times 300 \times 6,520.8 = 1,174 \text{ kN} \)

\[ \phi_{V_z} = 1,056 \text{ kN} \]

**Check for Axial Compression**

Section Compression Capacity

The flange slenderness, \( \lambda_{eb} = 6.98 \) [Ref : Cl no - 6.2.3.1]

Yield slenderness for flange, \( \lambda_{eby} = 16.00 \) [Ref : Table 6.2.4]

The web slenderness, \( \lambda_{ew} = 55.66 \)

Yield slenderness for web, \( \lambda_{eby} = 45 \)

Effective Width, \( b_e = 209 \text{ mm} \)

Effective Depth, \( DE = 405.7 \text{ mm} \)

Gross Area, \( A_g = 11,800 \text{ mm}^2 \)
Net Area, \( A_n = 10,820 \, \text{mm}^2 \)

Form factor, \( K_f = \frac{A_e}{A_g} = 0.917 \)

The nominal member section capacity for axial compression, 
\[ N_s = K_f \times A_n \times f_y = 0.917 \times 10,819.8 \times 300 = 3,246 \, \text{kN} \]
\[ \phi N_s = 0.9 \times 3,246 = 2,920 \, \text{kN} \]  
[Ref : Cl no - 6.2.1]

**Member Compression Capacity**

Effective length factor for slenderness & buckling about minor Y-axis, \( K_y = 1.00 \)

Effective length factor for slenderness & buckling about minor Z-axis, \( K_z = 1.00 \)

Effective Length of member, \( L_{ez} = 15,000 \, \text{mm} \)

Effective Length of member, \( L_{ey} = 15,000 \, \text{mm} \)

Geometrical Slenderness Ratio = 69.23

Geometrical Slenderness Ratio = 334.0

Member slenderness, \( \lambda_{nz} = \frac{L_{ez}}{r} \times \sqrt{K_f} \times \sqrt{f_y/250} \)  
\[ \lambda_{nz} = 69.23 \times \sqrt{1} \times \sqrt{300/250} = 72.62 \]  
[Ref : Cl no - 6.3.3.]

Member slenderness, \( \lambda_{ny} = \frac{L_{ey}}{r} \times \sqrt{K_f} \times \sqrt{f_y/250} \)  
\[ \lambda_{ny} = 334 \times \sqrt{1} \times \sqrt{300/250} = 350.35 \]  
[Ref : Cl no - 6.3.3]

\[ a_{az} = (\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2,050) = 19.984 \]

\[ a_{ay} = (\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2,050) = 5.923 \]

\[ a_b = 0.00 \]  
[Ref : table 6.3.3(2)]

\[ \lambda_z = \lambda_{nz} + a_{az}a_b = 72.62 \]

\[ \lambda_y = \lambda_{ny} + a_{ay}a_b = 350.35 \]

\[ \eta = 0.19 \]

\[ \eta = 1.10 \]

\[ \xi_z = ((\lambda_z/90)2 + 1 + \eta)/(2 \times (\lambda_z/90)2) = 1.42 \]

\[ \xi_y = ((\lambda_y/90)2 + 1 + \eta)/(2 \times (\lambda_y/90)2) = 0.57 \]

\[ a_{cz} = 0.731 \]  
[Ref : Cl no - 6.3.3]

\[ a_{cy} = 0.061 \]  
[Ref : Cl no - 6.3.3]

The nominal member capacity, \( N_{cz} = a_{cz} \times N_s \)  
\[ N_{cz} = a_{cz} \times N_s = 0.731 \times 3,246 = 2,373 \, \text{kN} \]

\[ \phi N_{cz} = 2,136 \, \text{kN} \]

The nominal member capacity, \( N_{cy} = a_{cy} \times N_s \)  
\[ N_{cy} = a_{cy} \times N_s = 0.061 \times 3,246 = 198.9 \, \text{kN} \]

\[ \phi N_{cy} = 178.96 \, \text{kN} \]

**Nominal Section Tension Capacity**

Ref. Cl 7.1
\[ K_t = 1.00 \]
\[ N_{t1} = A_g \times f_y = 3,540 \text{kN} \]
\[ N_{t2} = 0.85 \times K_t \times A_n \times f_u = 4,413 \text{kN} \]
\[ \phi_{Nt} = 3,186 \text{kN} \] [Ref: Cl no - 5.6.1.1.1(a)]

**Results**

Table 527: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \phi_{Msz} (\text{kN} \cdot \text{m}) )</td>
<td>639.9</td>
<td>639.9001</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Msy} (\text{kN} \cdot \text{m}) )</td>
<td>92.24</td>
<td>92.2392</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Mbz} (\text{kN} \cdot \text{m}) )</td>
<td>251.2</td>
<td>251.3427</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Vy} (\text{kN}) )</td>
<td>939.4</td>
<td>939.4444</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Vz} (\text{kN}) )</td>
<td>1,056</td>
<td>1,056.4</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Ns} (\text{kN}) )</td>
<td>2,920</td>
<td>2,921.3</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Ncz} (\text{kN}) )</td>
<td>2,136</td>
<td>2,136</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Ncy} (\text{kN}) )</td>
<td>178.96</td>
<td>179.0</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( \phi_{Nt} (\text{kN}) )</td>
<td>3,186</td>
<td>3,186.0</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD.Pro Input File**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS 3404 1997\NZS3404 1997-UB Section.std is typically installed with the program.

STAAD SPACE
*  INPUT FILE: NZS3404_Frame.STD
*  REFERENCE : Hand Calculation
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF UB,UC SHAPE PER
  THE NZS3404-1997 CODE
*  START JOB INFORMATION
ENGINEER DATE 16-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
*  UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 10 5 0; 6 10 0 0; 7 0 10 0; 8 5 10 0;
MEMBER INCIDENCES

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
</table>
| 1 | 1 | 2 | 2
| 2 | 1 | 1 | 2
| 3 | 3 | 4 | 4
| 4 | 3 | 5 | 5
| 5 | 5 | 6 | 6
| 6 | 2 | 7 | 7
| 7 | 3 | 8 | 8
| 8 | 5 | 9 | 9
| 9 | 7 | 8 | 8
| 10| 1 | 1 | 2
| 12| 8 | 11| 11
| 13| 9 | 12| 12
| 14| 10| 11| 11
| 15| 11| 12| 12
| 16| 2 | 14| 14
| 17| 3 | 15| 15
| 18| 5 | 17| 17
| 19| 7 | 19| 19
| 20| 8 | 20| 20
| 21| 9 | 21| 21
| 22| 10| 22| 22
| 23| 11| 23| 23
| 24| 12| 24| 24
| 25| 13| 14| 14
| 26| 14| 15| 15
| 27| 15| 16| 16
| 28| 15| 17| 17
| 29| 17| 18| 18
| 30| 14| 19| 19
| 31| 15| 20| 20
| 32| 17| 21| 21
| 33| 19| 20| 20
| 34| 20| 21| 21
| 35| 19| 22| 22
| 36| 20| 23| 23
| 37| 21| 24| 24
| 38| 22| 23| 23
| 39| 23| 24| 24
| 40| 14| 26| 26
| 41| 15| 27| 27
| 42| 17| 29| 29
| 43| 19| 31| 31
| 44| 20| 32| 32
| 45| 21| 33| 33
| 46| 22| 34| 34
| 47| 23| 35| 35
| 48| 24| 36| 36
| 49| 25| 36| 36
| 50| 26| 27| 27
| 51| 27| 28| 28
| 52| 27| 29| 29
| 53| 29| 30| 30
| 54| 26| 31| 31
| 55| 27| 32| 32
| 56| 29| 33| 33
| 57| 31| 34| 34
| 58| 32| 35| 35
| 59| 31| 34| 34
| 60| 32| 35| 35
| 61| 33| 36| 36
| 62| 34| 35| 35
| 63| 35| 36| 36
| 64| 26| 38| 38
| 65| 27| 39| 39
| 66| 29| 41| 41
| 67| 31| 43| 43
| 68| 32| 44| 44
| 69| 33| 45| 45
| 70| 34| 46| 46
| 71| 35| 47| 47
| 72| 36| 48| 48
| 73| 37| 38| 38
| 74| 38| 39| 39
| 75| 39| 40| 40
| 76| 39| 41| 41
| 77| 41| 42| 42
| 78| 38| 43| 43
| 79| 39| 44| 44
| 80| 41| 45| 45
| 81| 43| 44| 44
| 82| 44| 45| 45
| 83| 43| 46| 46
| 84| 44| 47| 47
| 85| 45| 48| 48
| 86| 46| 47| 47
| 87| 47| 48| 48
| 88| 44| 49| 49|

DEFINE PMEMBER

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
</table>
| 22| 46| 70| PMEMBER 1
| 19| 43| 67| PMEMBER 2
| 16| 40| 64| PMEMBER 3
| 20| 44| 68| PMEMBER 4
| 17| 41| 65| PMEMBER 5
| 24| 48| 72| PMEMBER 6
| 21| 45| 69| PMEMBER 7
| 18| 42| 66| PMEMBER 8
| 1| 6| 11| PMEMBER 9
| 25| 30| 35| PMEMBER 10
| 49| 54| 59| PMEMBER 11
| 73| 78| 83| PMEMBER 12
| 88| PMEMBER 13

DEFINE MATERIAL START

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>ISOTROPIC STEEL</td>
<td></td>
</tr>
<tr>
<td>E 2.05e+08</td>
<td></td>
</tr>
<tr>
<td>POISSON 0.3</td>
<td></td>
</tr>
<tr>
<td>DENSITY 76.8195</td>
<td></td>
</tr>
<tr>
<td>ALPHA 1.2e-05</td>
<td></td>
</tr>
<tr>
<td>DAMP 0.03</td>
<td></td>
</tr>
<tr>
<td>TYPE STEEL</td>
<td></td>
</tr>
<tr>
<td>STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2</td>
<td></td>
</tr>
</tbody>
</table>

END DEFINE MATERIAL

MEMBER PROPERTY AUSTRALIAN

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>4</td>
<td>9</td>
<td>10</td>
</tr>
<tr>
<td>14</td>
<td>TO</td>
<td>24</td>
<td>26</td>
</tr>
<tr>
<td>28</td>
<td>33</td>
<td>34</td>
<td>38</td>
</tr>
<tr>
<td>38</td>
<td>TO</td>
<td>48</td>
<td>50</td>
</tr>
<tr>
<td>52</td>
<td>57</td>
<td>58</td>
<td>62</td>
</tr>
<tr>
<td>TO</td>
<td>72</td>
<td>74</td>
<td>76</td>
</tr>
<tr>
<td>81</td>
<td>82</td>
<td>86</td>
<td>86</td>
</tr>
<tr>
<td>87</td>
<td>TO</td>
<td>88</td>
<td>TABLE ST UB530X92.4</td>
</tr>
</tbody>
</table>
Verification Examples
V.09 Steel Design

1 3 5 TO 8 11 TO 13 25 27 29 TO 32 35 TO 37 49 51 53 TO 56 59 TO 61 73 75 -
77 TO 80 83 TO 85 TABLE ST UC310X158
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 4 6 13 16 18 25 28 30 37 40 42 FIXED
*
MEMBER RELEASE
2 4 9 10 14 TO 24 26 28 33 34 38 39 50 52 57 58 62 63 74 76 81 82 86 -
87 START MY MZ
2 4 9 10 14 15 26 28 33 34 38 39 50 52 57 58 62 TO 66 68 69 71 72 74 76 81 -
82 86 87 END MY MZ
*
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
SELFWEIGHT Y -1
JOINT LOAD
23 49 FY -200
46 49 FX 200
36 49 FZ -200
MEMBER LOAD
20 44 68 88 UNI Y -50
26 88 CMOM GZ -30
88 CON GZ 20
*
PERFORM ANALYSIS
*
PRINT ANALYSIS RESULTS
*
PRINT MEMBER FORCES
*
PARAMETER 1
CODE NZS3404 1997
BEAM 1 PMEMB 5 13
DMAX 1.5 PMEMB 5 13
DMIN 0.4 PMEMB 5 13
IST 2 PMEMB 5 13
LHT 1 PMEMB 5 13
NSC 1 PMEMB 5 13
NSF 1 PMEMB 5 13
RATIO 1 PMEMB 5 13
SGR 0 PMEMB 5 13
SKL 1 PMEMB 5 13
SKR 1 PMEMB 5 13
SKT 1 PMEMB 5 13
TMAIN 400 PMEMB 5 13
TRACK 2 PMEMB 5 13
TSP 0 PMEMB 5 13
DUCT 1 PMEMB 5
GLD 1 PMEMB 5
CHECK CODE PMEMB 5
*
FINISH
**STAAD.Pro Output**

**STEEL DESIGN**

NOTE: SGR NOT SPECIFIED OR "DEFAULT" SPECIFIED FOR PMEMBER NO. 5.

NOTE: BY DEFAULT "AS/NZS 3679.1 300" WILL BE USED FOR ROLLED SECTIONS.

STAAD SPACE -- PAGE NO. 21

* 

STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)

*******************************************************************************

AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-

<table>
<thead>
<tr>
<th>STAAD.Pro</th>
<th>NZS3404 Spec.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X/x</td>
<td>Z/z</td>
<td>Longitudinal axis of section</td>
</tr>
<tr>
<td>Y/y</td>
<td>Y/y</td>
<td>Minor principal axis of section</td>
</tr>
<tr>
<td>Z/z</td>
<td>X/x</td>
<td>Major Principal axis of section</td>
</tr>
</tbody>
</table>

MEMBER DESIGN OUTPUT FOR PMEMBER 5

DESIGN Notes

1. (*) next to a Load Case number signifies that a P-Delta analysis has not been performed for that particular Load Case; i.e. analysis does not include second-order effects.

2. \( f = 0.9 \) for all the calculations [NZS3404 Table 3.4]

3. (#) next to Young's modulus \( E \) indicates that its value is not 200000 MPa as per NZS3404 1.4.

**DESIGN SUMMARY**

---------------

Designation: ST UB530X92.4 (AISC SECTIONS)

Governing Load Case: 1*

Governing Criteria: Cl. 12.8.3.1.1

Governing Ratio: 14.910 *(FAIL)

Governing Location: 10.015 m from Start.

**SECTION PROPERTIES**

--------------

d: 532.9999 mm  
bf: 209.0000 mm  

bf: 15.6000 mm  
tw: 10.2000 mm  

Ag: 11800.0000 mm²  
J: 775.0001E+03 mm⁴  
Iw: 1.5886E+12 mm⁶  

Iz: 554.0001E+06 mm⁴  
Sy: 355.0000E+03 mm³ (plastic)  
Zy: 227.7512E+03 mm³ (elastic)  

rz: 216.6775E+00 mm  
ry: 44.9105E+00 mm  

**STAAD SPACE** -- PAGE NO. 22

* 

**MATERIAL PROPERTIES**

-------------------

Material Standard: AS/NZS 3679.1

Nominal Grade: 300

Residual Stress Category: HR (Hot-rolled)

E (#): 204999.984 MPa [NZS3404 1.4]

G: 80000.000 MPa [NZS3404 1.4]

fy, flange: 300.000 MPa [NZS3404 Table 2.1]

fy, web: 320.000 MPa [NZS3404 Table 2.1]
<table>
<thead>
<tr>
<th><strong>BENDING</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Section Bending Capacity (about Z-axis)</strong></td>
</tr>
<tr>
<td>Critical Load Case :</td>
</tr>
<tr>
<td>Critical Ratio :</td>
</tr>
<tr>
<td>Critical Location :</td>
</tr>
<tr>
<td>$M_z^* = 164.5103E+00$ KNm</td>
</tr>
<tr>
<td>Section Slenderness: Compact</td>
</tr>
<tr>
<td>$Z_e = 2.3700E+06$ mm³</td>
</tr>
<tr>
<td>$f_{Ms} = 639.9001E+00$ KNm</td>
</tr>
<tr>
<td><strong>Section Bending Capacity (about Y-axis)</strong></td>
</tr>
<tr>
<td>Critical Load Case :</td>
</tr>
<tr>
<td>Critical Ratio :</td>
</tr>
<tr>
<td>Critical Location :</td>
</tr>
<tr>
<td>$M_y^* = -26.8162E+00$ KNm</td>
</tr>
<tr>
<td>Section Slenderness: Compact</td>
</tr>
<tr>
<td>$Z_e = 341.6269E+03$ mm³</td>
</tr>
<tr>
<td>$f_{Ms} = 92.2392E+00$ KNm</td>
</tr>
</tbody>
</table>

| **Member Bending Capacity** |
| Critical Load Case : | 1* |
| Critical Ratio : | 0.655 |
| Critical Location : | 5.015 m from Start. |
| Critical Flange Segment: |
| Location (Type): | 0.00 m(F )- 15.00 m(F ) |
| $M_z^* = 164.5103E+00$ KNm |
| $k_t = 1.00$ | [NZS3404 Table 5.6.3(1)] |
| $k_l = 1.00$ | [NZS3404 Table 5.6.3(2)] |
| $k_r = 1.00$ | [NZS3404 Table 5.6.3(3)] |
| $l_e = 15.00$ m | [NZS3404 5.6.3] |
| $a_m = 2.500$ | [NZS3404 5.6.1.1.1(b)(iii)] |
| $M_o = 127.0231E+00$ KNm | [NZS3404 5.6.1.1.1(d)] |
| $a_{sz} = 0.157$ | [NZS3404 5.6.1.1.1(c)] |
| $f_{Mb} = 251.3427E+00$ KNm $\leq f_{Ms}$ | [NZS3404 5.6.1.1.1(a)] |

<table>
<thead>
<tr>
<th><strong>SHEAR</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Section Shear Capacity (along Y-axis)</strong></td>
</tr>
<tr>
<td>Critical Load Case :</td>
</tr>
<tr>
<td>Critical Ratio :</td>
</tr>
<tr>
<td>Critical Location :</td>
</tr>
<tr>
<td>$V_y^* = 66.8261E+00$ KN</td>
</tr>
<tr>
<td>$f_{Vyy} = 939.4444E+00$ KN</td>
</tr>
<tr>
<td><strong>Section Shear Capacity (along Z-axis)</strong></td>
</tr>
<tr>
<td>Critical Load Case :</td>
</tr>
<tr>
<td>Critical Ratio :</td>
</tr>
<tr>
<td>Critical Location :</td>
</tr>
<tr>
<td>$V_z^* = 2.4943E+00$ KN</td>
</tr>
<tr>
<td>$f_{Vyz} = 1.0564E+03$ KN</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>AXIAL</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Section Compression Capacity</strong></td>
</tr>
<tr>
<td>Critical Load Case :</td>
</tr>
</tbody>
</table>
Critical Ratio : 0.039
Critical Location : 10.015 m from Start.
N* = 114.4841E+00 KN
Ae = 10.8198E+03 mm² [NZS3404 6.2.3 / 6.2.4]
kf = 0.917 [AS 4100 6.2.2]
An = 11.8000E+03 mm²
f Ns = 2.9213E+03 KN [NZS3404 6.2.1]

Member Compression Capacity (about Z-axis)
Critical Load Case : 1*
Critical Ratio : 0.054
Critical Location : 10.015 m from Start.
N* = 114.4841E+00 KN

Unbraced Segment:
Location (Type): 0.00 m(U) - 15.00 m(U)
Lez = 15.00 m

Critical Ratio : 0.54
Critical Location : 10.015 m from Start.
N* = 114.4841E+00 KN

Unbraced Segment:
Location (Type): 0.00 m(U) - 15.00 m(U)
Lez = 15.00 m

Critical Ratio : 0.640
Critical Location : 10.015 m from Start.
N* = 114.4841E+00 KN

Unbraced Segment:
Location (Type): 0.00 m(U) - 15.00 m(U)

Critical Ratio : 0.000
Critical Location : 0.000 m from Start.
N* = 0.0000E+00 KN

kt = 1.00 [User defined]
An = 11.8000E+03 mm²
f Nt = 3.1860E+03 KN [NZS3404 7.2]

Section Tension Capacity

Critical Load Case : 1*
Critical Ratio : 0.000
Critical Location : 0.000 m from Start.
N* = 0.0000E+00 KN

COMBINED BENDING AND AXIAL

Section Combined Capacity (about Z-axis)
Critical Load Case : 1*
Critical Ratio : 0.257
Critical Location : 5.015 m from Start.
f Mrz = 639.9001E+00 KNm [NZS3404 8.3.2]
Section Combined Capacity (about Y-axis)
Critical Load Case : 1*
Critical Ratio : 0.291
Critical Location : 10.015 m from Start.
f Mry = 92.2392E+00 KNm [NZS3404 8.3.3]
Section Combined Capacity (Biaxial)
Critical Load Case : 1*
Critical Ratio : 0.271
Critical Location : 9.985 m from Start.
\[ g = 1.426 \] [NZS3404 8.3.4]

Member In-plane Capacity (about Z-axis)
Critical Load Case : 1*
Critical Ratio : 0.267
Critical Location : 5.015 m from Start.
\[ f_{Miz} = 616.9343 \times 10^0 \text{ KNm} \] [NZS3404 8.4.2]

Member In-plane Capacity (about Y-axis)
Critical Load Case : 1*
Critical Ratio : 0.806
Critical Location : 10.015 m from Start.
\[ f_{Miy} = 33.2658 \times 10^0 \text{ KNm} \] [NZS3404 8.4.2]

Member Out-of-plane Capacity (Tension)
Critical Load Case : 1*
Critical Ratio : 0.000
Critical Location : 0.000 m from Start.
\[ a_{bc} = 0 \]
\[ f_{Noz} = 0 \times 10^0 \text{ KN} \] [NZS3404 8.4.4.1.2]
\[ f_{Moz,t} = 0 \times 10^0 \text{ KNm} \] [NZS3404 8.4.4.1]

Member Out-of-plane Capacity (Compression)
Critical Load Case : 1*
Critical Ratio : 1.150
Critical Location : 5.015 m from Start.
\[ f_{Moz,c} = 143.1029 \times 10^0 \text{ KNm} \] [NZS3404 8.4.4.2]

Member Biaxial Capacity (Tension)
Critical Load Case : 1*
Critical Ratio : 0.000
Critical Location : 0.000 m from Start.

Member Biaxial Capacity (Compression)
Critical Load Case : 1*
Critical Ratio : 1.918
Critical Location : 10.015 m from Start.

Section Slenderness (Bending about Z-axis)
Critical Load Case : 1*
Critical Ratio : 0.776
Critical Location : 0.000 m from Start.
\[ l_{sz} = 6.98 \] [NZS3404 12.5.1.1]
\[ l_{ez} = 9.00 \] [NZS3404 Table 12.5]

Section Slenderness (Bending about Y-axis)
Critical Load Case : 1*
Critical Ratio : 0.776
Critical Location : 0.000 m from Start.
\[ l_{sy} = 6.98 \] [NZS3404 12.5.1.1]
\[ l_{ey} = 9.00 \] [NZS3404 Table 12.5]

Max Specific Yield Stress
Critical Load Case : 1*
Critical Ratio : 0.833
Critical Location : 0.000 m from Start.
\[ F_{Y, actual} = 300.00 \]
\[ F_{Y, limit} = 360.00 \] [NZS3404 Table 12.4(1)]
Max Actual Yield Ratio (Fy/Fu)
Critical Load Case : 1*
Critical Ratio : 0.852
Critical Location : 0.000 m from Start.
Fy/Fu,actual = 0.68
Fy/Fu,limit = 0.80 [NZS3404 Table 12.4(3)]

Fabrication Requirement
Critical Load Case : N/A
Critical Ratio : N/A
Critical Location : N/A
Status = Passed [NZS3404 12.4.1.2]

Section Symmetry Requirement
Critical Load Case : N/A
Critical Ratio : N/A
Critical Location : N/A
Status = Passed [NZS3404 12.5.2]

Min Web Thickness Requirement for Beam
Critical Load Case : 1*
Critical Ratio : 0.679
Critical Location : 0.000 m from Start.
tw,actual = 10.20
tw,min = 6.92 [NZS3404 12.7.2]

Max Axial Force Limit for Column (a)
Critical Load Case : 1*
Critical Ratio : 0.078
Critical Location : 10.015 m from Start.
N*/f Ns - actual = 0.04
N*/f Ns - limit = 0.50 [NZS3404 Table 12.8.1]

Max Axial Force Limit for Column (b)
Critical Load Case : 1*
Critical Ratio : 14.910
Critical Location : 10.015 m from Start.
b m = 0.00
NoL = 214.0169E+00 KN
l EYC = 3.89
N*/f Ns - actual = 0.04
N*/f Ns - limit = 0.00 [NZS3404 12.8.3.1(b)]

Max Axial Force Limit for Column (c)
Critical Load Case : 1*
Critical Ratio : 0.202
Critical Location : 10.015 m from Start.
Ng*/f Ns - actual = 0.04
Ng*/f Ns - limit = 0.19 [NZS3404 12.8.3.1(c)]

Shear-Y + Bend-Z Interaction
Critical Load Case : 1*
Critical Ratio : 0.253
Critical Location : 5.015 m from Start.
Mz* = -161.9652E+00 KN
f Msvz = 639.9001E+00 KN [NZS3404 12.10.3.1]

Shear-Z + Bend-Y Interaction
Critical Load Case : 1*
Critical Ratio : 0.151
Critical Location : 9.985 m from Start.
My* = -13.9665E+00 KN
f Msvy = 92.2392E+00 KN [NZS3404 12.10.3.1]
V. United Kingdom

V. Steel Design per BS 5950-1:2000

V. BS5950 2000 - Fully Restrained Simply Supported Beam

A 6.5 m, simply-supported beam is fully restrained along its length. The beam is designed in S275 steel for the loading described.

Reference


Problem

The beam is designed in S275 steel for the loading described.

Dead Loads, $\gamma_{fd} = 1.4$:
- Distributed load (including s/w) $w_d = 15 \text{ kN/m}$
- Point Load $W_d = 40 \text{ kN}$

Imposed Loads, $\gamma_{fi} = 1.6$:
- Distributed load $W_i = 30 \text{ kN/m}$
- Point Load $W_i = 50 \text{ kN}$

Comparison

Table 528: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear capacity, $P_v$ (kN)</td>
<td>888</td>
<td>888.4</td>
<td>none</td>
</tr>
<tr>
<td>Moment capacity, $M_{cx}$ (kN-m)</td>
<td>649</td>
<td>649</td>
<td>none</td>
</tr>
<tr>
<td>Total deflection, $\delta$ (mm)</td>
<td>8.69</td>
<td>9.25$^a$</td>
<td>6.4%</td>
</tr>
<tr>
<td>Deflection limit (mm)</td>
<td>18.1</td>
<td>18.056</td>
<td>none</td>
</tr>
</tbody>
</table>
a. STAAD.Pro includes effects of shear deformation, resulting in a higher calculated maximum deflection.

**STAAD Input**

| TRACK 2 ALL | Maximum detail output |
| UNI 0 ALL | Identifies the beam as fully restrained |
| DEF 360 ALL | Limiting ratio of beam length to maximum deflection |
| DJ1 1 ALL | Identifies starting joint of "physical member" for deflection check |
| DJ2 3 ALL | Identifies ending joint of "physical member" for deflection check |

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Fully Restrained Simply Supported Beam.STD is typically installed with the program.

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Fully Restrained Simply Supported Beam.STD is typically installed with the program.

**STAAD SPACE**

**START JOB INFORMATION**

**JOB NAME** Example no. 2

**JOB CLIENT** The Steel Construction Institute

**JOB COMMENT** Simply supported restrained beam

**ENGINEER DATE** Jun-2003

**END JOB INFORMATION**

**INPUT WIDTH** 79

*********************************************

**UNIT METER KN**

**JOINT COORDINATES**

1 0 0 0; 2 3.25 0 0; 3 6.5 0 0;

**MEMBER INCIDENCES**

1 1 2; 2 2 3;

**DEFINE MATERIAL START**

**ISOTROPIC STEEL**

**E** 2.05e+08

**POISSON** 0.3

**DENSITY** 76.8195

**ALPHA** 1.2e-05

**DAMP** 0.03

**END DEFINE MATERIAL**

**MEMBER PROPERTY BRITISH**

1 2 TABLE ST UB533X210X92

**CONSTANTS**

**MATERIAL STEEL ALL**

**SUPPORTS**

1 3 FIXED BUT MZ

*********************************************

**Loading**

**LOAD 1 DEAD**

**JOINT LOAD**

2 FY -40

**MEMBER LOAD**

1 2 UNI GY -15
LOAD 2 LIVE
JOINT LOAD
2 FY -50
MEMBER LOAD
1 2 UNI GY -30

LOAD COMB 100 COMBINATION LOAD CASE 3
1 1.4 2 1.6
PERFORM ANALYSIS PRINT STATICS CHECK

* First check the forces
PARAMETER 1
CODE BS5950
TRACK 2 ALL
UNL 0 ALL
CHECK CODE ALL

* Second check the displacements
UNIT MMS KN
LOAD LIST 2
PARAMETER 2
CODE BS5950
DFF 360 ALL
DJ1 1 ALL
DJ2 3 ALL
TRACK 4 ALL
CHECK CODE ALL
FINISH

***********************************************************************

STAAD Output

PAGE NO.

1

************************************************************************

*  STAAD.Pro CONNECT Edition  
*  Version 22.01.00.**  
*  Proprietary Program of  
*  Bentley Systems, Inc.  
*  Date= APR 14, 2019  
*  Time= 23:10:28  
*  Licensed to: Bentley Systems Inc  
************************************************************************

1. STAAD SPACE
INPUT FILE: BS5950 2000 - Fully Restrained Simply Supported Beam.STD
2. START JOB INFORMATION
3. JOB NAME EXAMPLE NO. 2
4. JOB CLIENT THE STEEL CONSTRUCTION INSTITUTE
5. JOB COMMENT SIMPLY SUPPORTED RESTRAINED BEAM
6. ENGINEER DATE JUN-2003
7. END JOB INFORMATION
8. INPUT WIDTH 79
9. *********************************************
10. UNIT METER KN
11. JOINT COORDINATES

***
12. 1 0 0 0; 2 3.25 0 0; 3 6.5 0 0
13. MEMBER INCIDENCES
14. 1 1 2; 2 2 3
15. DEFINE MATERIAL START
16. ISOTROPIC STEEL
17. E 2.05E+08
18. POISSON 0.3
19. DENSITY 76.8195
20. ALPHA 1.2E-05
21. DAMP 0.03
22. END DEFINE MATERIAL
23. MEMBER PROPERTY BRITISH
24. 1 2 TABLE ST UBS33X210X92
25. CONSTANTS
26. MATERIAL STEEL ALL
27. SUPPORTS
28. 1 3 FIXED BUT MZ
29. ****************************************************
30. * LOADING
31. LOAD 1 DEAD
32. JOINT LOAD
33. 2 FY -40
34. MEMBER LOAD
35. 1 2 UNI GY -15
36. ****************************************************
37. LOAD 2 LIVE
38. JOINT LOAD
39. STAAD SPACE
40. 2 2 FY -50
41. MEMBER LOAD
42. 1 2 UNI GY -30
43. ****************************************************
44. LOAD COMB 100 COMBINATION LOAD CASE 3
45. 1 1.4 2 1.6
46. PERFORM ANALYSIS PRINT STATICS CHECK

Problem Statistics
-----------------------------------
NUMBER OF JOINTS          3  NUMBER OF MEMBERS       2
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       8
TOTAL LOAD COMBINATION CASES =     1  SO FAR.

Static Load/Reaction/Equilibrium Summary for Case No. 1
DEAD
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.325000013E+01
Y =  0.000000000E+00
Z =  0.000000000E+00
TOTAL APPLIED LOAD 1
***TOTAL APPLIED LOAD ( KN   METE ) SUMMARY (LOADING 1 )
SUMMATION FORCE-X =        0.00
SUMMATION FORCE-Y =        -137.50
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=           0.00  MY=           0.00  MZ=        -446.88
TOTAL REACTION LOAD    1
***TOTAL REACTION LOAD( KN  METE ) SUMMARY (LOADING  1 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =         137.50
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=           0.00  MY=           0.00  MZ=        446.88
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING  1)
MAXIMUMS    AT NODE
X =  0.00000E+00       0
Y = -5.44310E-01       2
Z =  0.00000E+00       0
RX=  0.00000E+00       0
RY=  0.00000E+00       0
RZ=  2.45021E-03       3
STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
LIVE
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.325000000E+01
Y =  0.000000000E+00
Z =  0.000000000E+00
STAAD SPACE                                              -- PAGE NO. 4
TOTAL APPLIED LOAD     2
***TOTAL APPLIED LOAD ( KN  METE ) SUMMARY (LOADING  2 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =        -245.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=           0.00  MY=           0.00  MZ=        -796.25
TOTAL REACTION LOAD    2
***TOTAL REACTION LOAD( KN  METE ) SUMMARY (LOADING  2 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =         245.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=           0.00  MY=           0.00  MZ=         796.25
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING  2)
MAXIMUMS    AT NODE
X =  0.00000E+00       0
Y = -9.25457E-01       2
Z =  0.00000E+00       0
RX=  0.00000E+00       0
RY=  0.00000E+00       0
RZ=  4.20036E-03       3
************ END OF DATA FROM INTERNAL STORAGE ************
46. *********************************************
47. * FIRST CHECK THE FORCES
48. PARAMETER 1
49. CODE BS5950
50. TRACK 2 ALL
51. UNL 0 ALL
52. CHECK CODE ALL
STEEL DESIGN
MATERIAL DATA

Grade of steel = S 275
Modulus of elasticity = 205 kN/mm²
Design Strength (py) = 275 N/mm²

SECTION PROPERTIES (units - cm)

Member Length = 325.00
Gross Area = 117.00 Net Area = 117.00 Eff. Area = 117.00

Shear Area = 58.771 53.843

DESIGN DATA (units - kN,m) BS5950-1/2000

Section Class = PLASTIC

Moment Capacity = 649.00 94.2
Reduced Moment Capacity = 649.00 94.2
Shear Capacity = 969.7 888.4

BUCKLING CALCULATIONS (units - kN,m)

LTB Moment Capacity (kNm) and LTB Length (m): 649.00 0.00

LTB Coefficients & Associated Moments (kNm): mL = 1.00 mx = 0.00 my = 0.00 myx = 0.00

Mlt = 585.41 Mx = 0.00 My = 0.00 My = 0.00

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

Clause Ratio Load FX VY VZ MZ MY

BS-4.2.3-(Y) 0.329 100 - 292.3 - -

BS-4.3.6 0.902 100 - 292.3 585.41 -

Torsion and deflections have not been considered in the design.
Gross Area = 117.00  Net Area = 117.00  Eff. Area = 117.00

\[
\begin{align*}
\text{Moment of inertia} & : 55200.004 \quad 2390.000 \\
\text{Plastic modulus} & : 2360.000 \quad 355.000 \\
\text{Elastic modulus} & : 2070.906 \quad 228.380 \\
\text{Effective modulus} & : 2360.000 \quad 355.000 \\
\text{Shear Area} & : 58.771 \quad 53.843 \\
\end{align*}
\]

DESIGN DATA (units - kN,m)  BS5950-1/2000

<table>
<thead>
<tr>
<th>Section Class</th>
<th>PLASTIC</th>
</tr>
</thead>
</table>

\[
\begin{align*}
\text{Moment Capacity} & : 649.0 \quad 94.2 \\
\text{Reduced Moment Capacity} & : 649.0 \quad 94.2 \\
\text{Shear Capacity} & : 969.7 \quad 888.4 \\
\end{align*}
\]

BUCKLING CALCULATIONS (units - kN,m)

<table>
<thead>
<tr>
<th>(axis nomenclature as per design code)</th>
</tr>
</thead>
<tbody>
<tr>
<td>LTB Moment Capacity (kNm) and LTB Length (m): 649.00, 0.00</td>
</tr>
<tr>
<td>LTB Coefficients &amp; Associated Moments (kNm):</td>
</tr>
<tr>
<td>mLT = 1.00 : mx = 0.00 : my = 0.00 : myx = 0.00</td>
</tr>
<tr>
<td>Mlt = 585.41 : Mx = 0.00 : My = 0.00 : My = 0.00</td>
</tr>
</tbody>
</table>

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>BS-4.2.3-(Y)</td>
<td>0.329</td>
<td>100</td>
<td>-</td>
<td>292.3</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>BS-4.3.6</td>
<td>0.902</td>
<td>100</td>
<td>-</td>
<td>68.0</td>
<td>-</td>
<td>585.4</td>
<td>-</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.

****** END OF TABULATED RESULT OF DESIGN *******

---

7

************** END OF TABULATED RESULT OF DESIGN **************

53. ************************************************************

54. * SECOND CHECK THE DISPLACEMENTS

55. UNIT MMS KN

56. LOAD LIST 2

57. PARAMETER 2

58. CODE BS5950

59. DFF 360 ALL

60. DJ1 1 ALL

61. DJ2 3 ALL

62. TRACK 4 ALL

63. CHECK CODE ALL

STEEL DESIGN

STAAD.Pro CODE CHECKING - (BSI)

********************************************************************************

PROGRAM CODE REVISION V2.13_5950-1_2000

---

8

CLAUSE BS-2.5.1

CHECKS

LENGTH UNITS - MMS

MEMBER TABLE RESULT ACTUAL DEFL. DEFL.LEN/LOAD/ DEFL. LIMIT DFF

LOCATION

*******************************************************************************

1  ST UB533X210X92 PASS 9.255 18.056 6500.000
V. BS5950 2000 - Unrestrained Simply Supported Beam

Design of a 9 m, unrestrained beam with end moments.

Reference


Problem

Intermediate point loads are applied to the bottom flange. These do not provide restraint against lateral-torsional buckling. The beam is designed in S275 steel.

Dead Loads, $\gamma_{fd}$ = 1.4:

- Selfweight $w_s = 3 \text{ kN/m}$
- Point Load $W_{1d} = 40 \text{ kN}$
- Point Load $W_{2d} = 20 \text{ kN}$

Imposed Loads, $\gamma_{fi}$ = 1.6:

- Point Load $W_{1i} = 60 \text{ kN}$
- Point Load $W_{2i} = 30 \text{ kN}$
Comparison

Table 529: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear capacity, $P_v$ (kN)</td>
<td>636</td>
<td>635.9</td>
<td>none</td>
</tr>
<tr>
<td>Moment capacity, $M_{cx}$ (kN-m)</td>
<td>404</td>
<td>404.5</td>
<td>none</td>
</tr>
<tr>
<td>Lateral-torsional buckling capacity, $M_b$ (kN-m)</td>
<td>150</td>
<td>150.26</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

- **TRACK 2 ALL**: Maximum detail output
- **MLT 0.46 ALL**: Specifies the $m_{LT}$ value to use in the lateral-torsional buckling calculations
- **UNI 6.3 ALL**: Specifies the lateral-torsional buckling length to be used

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Unrestrained Simply Supported Beam.STD is typically installed with the program.
1 4 FIXED
*************************************
LOAD 1 DEAD
JOINT LOAD
2 FY -40
MEMBER LOAD
1 TO 3 UNI GY -3
UNIT MMS KN
JOINT LOAD
3 FY -20
UNIT METER KN
LOAD 2 LIVE
JOINT LOAD
2 FY -60
UNIT MMS KN
JOINT LOAD
3 FY -30
UNIT METER KN
LOAD COMB 100 COMBINATION LOAD CASE 3
1 1.4 2 1.6
PERFORM ANALYSIS PRINT STATICS CHECK
*************************************
PARAMETER 1
CODE BS5950
TRACK 2 ALL
MLT 0.46 ALL
UNL 6.3 ALL
CHECK CODE ALL
FINISH
*************************************

**STAAD Output**

1 INPUT FILE: BS5950 2000 - Unrestrained Simply Supported Beam.STD
2 START JOB INFORMATION
3 JOB NAME EXAMPLE NO. 3
4 JOB CLIENT THE STEEL CONSTRUCTION INSTITUTE
5 JOB COMMENT UNRESTRAINED BEAM WITH END MOMENTS
6 ENGINEER DATE JUN-2003
7 END JOB INFORMATION
8 INPUT WIDTH 79
9 UNIT METER KN
10 JOINT COORDINATES

**STAAD.Pro CONNECT Edition**
Version 22.01.00.**
Proprietary Program of Bentley Systems, Inc.
Date= APR 14, 2019
Time= 23:10:37
Licensed to: Bentley Systems Inc
11. 1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0
12. MEMBER INCIDENCES
13. 1 1 2; 2 2 3; 3 3 4
14. *************************************
15. DEFINE MATERIAL START
16. ISOTROPIC STEEL
17. E 2.05E+08
18. POISSON 0.3
19. DENSITY 76.8195
20. ALPHA 1.2E-05
21. DAMP 0.03
22. END DEFINE MATERIAL
23. MEMBER PROPERTY BRITISH
24. 1 TO 3 TABLE ST UB457X191X67
25. CONSTANTS
26. MATERIAL STEEL ALL
27. SUPPORTS
28. 1 4 FIXED
29. *************************************
30. LOAD 1 DEAD
31. JOINT LOAD
32. 2 FY -40
33. MEMBER LOAD
34. 1 TO 3 UNI GY -3
35. UNIT MMS KN
36. JOINT LOAD
37. 3 FY -20
38. UNIT METER KN
39. STAAD SPACE

2
39. LOAD 2 LIVE
40. JOINT LOAD
41. 2 FY -60
42. UNIT MMS KN
43. JOINT LOAD
44. 3 FY -30
45. UNIT METER KN
46. LOAD COMB 100 COMBINATION LOAD CASE 3
47. 1 1.4 2 1.6
48. PERFORM ANALYSIS PRINT STATICS CHECK

PROBLEM STATISTICS

- NUMBER OF JOINTS = 4
- NUMBER OF MEMBERS = 3
- NUMBER OF PLATES = 0
- NUMBER OF SOLIDS = 0
- NUMBER OF SURFACES = 0
- NUMBER OF SUPPORTS = 2

Using 64-bit analysis engine.

SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

- PRIMARY LOAD CASES = 2
- TOTAL DEGREES OF FREEDOM = 12
- TOTAL LOAD COMBINATION CASES = 1

STAAD SPACE

3

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 1

DEAD CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)

   X = 0.415517241E+01
   Y = 0.00000000E+00
   Z = 0.00000000E+00
TOTAL APPLIED LOAD 1
***TOTAL APPLIED LOAD ( KN METE ) SUMMARY (LOADING 1 )
  SUMMATION FORCE-X = 0.00
  SUMMATION FORCE-Y = -87.00
  SUMMATION FORCE-Z = 0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX= 0.00 MY= 0.00 MZ= -361.50
TOTAL REACTION LOAD 1
***TOTAL REACTION LOAD( KN METE ) SUMMARY (LOADING 1 )
  SUMMATION FORCE-X = 0.00
  SUMMATION FORCE-Y = 87.00
  SUMMATION FORCE-Z = 0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX= 0.00 MY= 0.00 MZ= 361.50
MAXIMUM DISPLACEMENTS ( CM /RADIANS ) (LOADING 1)
  MAXIMUMS AT NODE
  X = 0.00000E+00 0
  Y = -3.47127E-01 2
  Z = 0.00000E+00 0
  RX= 0.00000E+00 0
  RY= 0.00000E+00 0
  RZ= 1.00460E-03 3

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 2
LIVE CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
  X = 0.399999998E+01
  Y = 0.000000000E+00
  Z = 0.000000000E+00

STAAD SPACE -- PAGE NO. 4
TOTAL APPLIED LOAD 2
***TOTAL APPLIED LOAD ( KN METE ) SUMMARY (LOADING 2 )
  SUMMATION FORCE-X = 0.00
  SUMMATION FORCE-Y = -90.00
  SUMMATION FORCE-Z = 0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX= 0.00 MY= 0.00 MZ= -360.00
TOTAL REACTION LOAD 2
***TOTAL REACTION LOAD( KN METE ) SUMMARY (LOADING 2 )
  SUMMATION FORCE-X = 0.00
  SUMMATION FORCE-Y = 90.00
  SUMMATION FORCE-Z = 0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX= 0.00 MY= 0.00 MZ= 360.00
MAXIMUM DISPLACEMENTS ( CM /RADIANS ) (LOADING 2)
  MAXIMUMS AT NODE
  X = 0.00000E+00 0
  Y = -4.06566E-01 2
  Z = 0.00000E+00 0
  RX= 0.00000E+00 0
  RY= 0.00000E+00 0
  RZ= 1.17091E-03 3

************ END OF DATA FROM INTERNAL STORAGE ************
49. ***************************************
50. PARAMETER 1
51. CODE BS5950
52. TRACK 2 ALL
### Verification Examples

#### V.09 Steel Design

53. MLT 0.46 ALL
54. UNL 6.3 ALL
55. CHECK CODE ALL

**STEEL DESIGN**

```
53. MLT 0.46 ALL
54. UNL 6.3 ALL
55. CHECK CODE ALL

PROGRAM CODE REVISION V2.13_5950-1_2000

---

ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>1 ST</td>
<td>UB457X191X67</td>
<td>PASS</td>
<td>BS-4.3.6</td>
<td>0.861</td>
<td>100</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>280.96</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**MATERIAL DATA**

- Grade of steel = S 275
- Modulus of elasticity = 205 kN/mm²
- Design Strength (py) = 275 N/mm²

**SECTION PROPERTIES (units - cm)**

- Member Length = 300.00
- Gross Area = 85.50
- Net Area = 85.50
- Eff. Area = 85.50

<table>
<thead>
<tr>
<th></th>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment of inertia</td>
<td>29400.002</td>
<td>1450.000</td>
</tr>
<tr>
<td>Plastic modulus</td>
<td>1470.000</td>
<td>237.000</td>
</tr>
<tr>
<td>Elastic modulus</td>
<td>1296.868</td>
<td>152.712</td>
</tr>
<tr>
<td>Effective modulus</td>
<td>1470.000</td>
<td>237.000</td>
</tr>
<tr>
<td>Shear Area</td>
<td>43.411</td>
<td>38.539</td>
</tr>
</tbody>
</table>

**DESIGN DATA (units - kN,m)**

- Section Class: PLASTIC
- Moment Capacity: 404.3, 63.0
- Reduced Moment Capacity: 404.3, 63.0
- Shear Capacity: 716.3, 635.9

**BUCKLING CALCULATIONS (units - kN,m)**

LTB Moment Capacity (kNm) and LTB Length (m): 150.11, 6.300
LTB Coefficients & Associated Moments (kNm):

- mLT = 0.46 : mx = 0.00 : my = 0.00 : myx = 0.00
- Mlt = 280.96 : Mx = 0.00 : My = 0.00 : My = 0.00

**CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):**

- BS-4.2.3-(Y) 0.238 100 - 151.0 - -
- BS-4.3.6 0.861 100 - 151.0 - 281.0 -

Torsion and deflections have not been considered in the design.

---

**STAAD SPACE**

---

### Verification Examples

#### V.09 Steel Design

53. MLT 0.46 ALL
54. UNL 6.3 ALL
55. CHECK CODE ALL

**STEEL DESIGN**

```
53. MLT 0.46 ALL
54. UNL 6.3 ALL
55. CHECK CODE ALL

PROGRAM CODE REVISION V2.13_5950-1_2000

---

ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>2 ST</td>
<td>UB457X191X67</td>
<td>PASS</td>
<td>BS-4.3.6</td>
<td>0.470</td>
<td>100</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>153.25</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**MATERIAL DATA**

- Grade of steel = S 275
Modulus of elasticity = 205 kN/mm²
Design Strength (py) = 275 N/mm²

SECTION PROPERTIES (units - cm)
Member Length = 300.00
Gross Area = 85.50 Net Area = 85.50 Eff. Area = 85.50

- z-z axis y-y axis

- Moment of inertia : 29400.002 1450.000
- Plastic modulus : 1470.000 237.000
- Elastic modulus : 1296.868 152.712
- Effective modulus : 1470.000 237.000
- Shear Area : 43.411 38.539

DESIGN DATA (units - kN,m) BS5950-1/2000
Section Class : PLASTIC

- z-z axis y-y axis

- Moment Capacity : 404.3 63.0
- Reduced Moment Capacity : 404.3 63.0
- Shear Capacity : 716.3 635.9

BUCKLING CALCULATIONS (units - kN,m)
(axis nomenclature as per design code)

- LTB Moment Capacity (kNm) and LTB Length (m): 150.11, 6.300
- LTB Coefficients & Associated Moments (kNm):
  - m_LT = 0.46 : m_X = 0.00 : m_Y = 0.00 : m_YX = 0.00
  - M_LT = 153.25 : M_X = 0.00 : M_Y = 0.00 : M_Y = 0.00

CRITICAL LOADS FOR EACH CLAUSE CHECK (units - kN,m):

- BS-4.2.3-(Y) 0.041 100 - 26.2 - - -
- BS-4.3.6 0.470 100 - 13.6 - 153.2 -

Torsion and deflections have not been considered in the design.

ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)

STAAD SPACE -- PAGE NO. 7

MATERIAL DATA
Grade of steel = S 275
Modulus of elasticity = 205 kN/mm²
Design Strength (py) = 275 N/mm²

SECTION PROPERTIES (units - cm)
Member Length = 300.00
Gross Area = 85.50 Net Area = 85.50 Eff. Area = 85.50

- z-z axis y-y axis

- Moment of inertia : 29400.002 1450.000
- Plastic modulus : 1470.000 237.000
- Elastic modulus : 1296.868 152.712
- Effective modulus : 1470.000 237.000
- Shear Area : 43.411 38.539

DESIGN DATA (units - kN,m) BS5950-1/2000
Section Class : PLASTIC

- z-z axis y-y axis

- Moment Capacity : 404.3 63.0
- Reduced Moment Capacity : 404.3 63.0
- Shear Capacity : 716.3 635.9

BUCKLING CALCULATIONS (units - kN,m)
### LTB Moment Capacity (kNm) and LTB Length (m):
- 150.11, 6.300

### LTB Coefficients & Associated Moments (kNm):
- \( mLT = 0.46 \)
- \( mX = 0.00 \)
- \( my = 0.00 \)
- \( myx = 0.00 \)
- \( Mlt = 231.74 \)
- \( Mx = 0.00 \)
- \( My = 0.00 \)
- \( My = 0.00 \)

### CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>BS-4.2.3-(Y)</td>
<td>0.180</td>
<td>100</td>
<td>-114.8</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>BS-4.3.6</td>
<td>0.710</td>
<td>100</td>
<td>-102.2</td>
<td>231.7</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.

---

### V. BS5950 2000 - Beam from UB Restrained at Loading

Design of a simply-supported, 9m beam, which is laterally restrained at the ends and at the points of load application only.

**Reference**


**Problem**

Dead Loads, \( \gamma_{fd} = 1.4 \):

- Selfweight \( w_s = 3 \text{ kN/m} \)
- Point Load \( W_{1d} = 40 \text{ kN} \)
- Point Load \( W_{2d} = 20 \text{ kN} \)

Imposed Loads, \( \gamma_{fi} = 1.6 \):

- Point Load \( W_{1i} = 60 \text{ kN} \)
- Point Load \( W_{2i} = 30 \text{ kN} \)
Comparison

Table 530: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear capacity, $P_v$ (kN)</td>
<td>751</td>
<td>751.4</td>
<td>none</td>
</tr>
<tr>
<td>Moment capacity, $M_{cx}$ (kN-m)</td>
<td>503</td>
<td>503.5</td>
<td>none</td>
</tr>
<tr>
<td>Lateral-torsional bucking moment, $M_b$ (kN-m)</td>
<td>397</td>
<td>396.6</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

- **TRACK 2 ALL**
- **MLT _MAIN J1 U2 U3 J4**

Maximum detail output

Specifies that $m_L$ is to be calculated with the end joints fully restrained and the intermediate joints restraining the upper flange

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Beam from UB Restrained at Loading.STD is typically installed with the program.

**STAAD SPACE**

**START JOB INFORMATION**

**JOB NAME** Example no. 4

**JOB CLIENT** The Steel Construction Institute

**JOB COMMENT** Simply supported beam with lateral restraint points

**ENGINEER DATE** Jun-2003

**END JOB INFORMATION**

**INPUT WIDTH** 79

**UNIT METER KN**

**JOINT COORDINATES**

1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0;

**MEMBER INCIDENCES**

1 1 2; 2 2 3; 3 3 4;

**START GROUP DEFINITION**

**MEMBER**

_ MAIN 1 TO 3

**END GROUP DEFINITION**

**DEFINE MATERIAL START**

**ISOTROPIC STEEL**

**E** 2.05e+08

**POISSON** 0.3

**DENSITY** 76.8195

**ALPHA** 1.2e-05

**DAMP** 0.03

**END DEFINE MATERIAL**
MEMBER PROPERTY BRITISH
1 TO 3 TABLE ST UB457X191X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED BUT MZ
*************************************
LOAD 1 DEAD
JOINT LOAD
2 FY -40
MEMBER LOAD
1 TO 3 UNI GY -3
UNIT MMS KN
JOINT LOAD
3 FY -20
UNIT METER KN
LOAD 2 LIVE
JOINT LOAD
2 FY -60
UNIT MMS KN
JOINT LOAD
3 FY -30
LOAD COMB 100 COMBINATION LOAD CASE 3
1 1.4 2 1.6
PERFORM ANALYSIS PRINT STATICS CHECK
*************************************
PARAMETER 1
CODE BS5950
TRACK 2 ALL
MLT _MAIN J1 U2 U3 J4
CHECK CODE ALL
FINISH
*************************************

STAAD Output

1

*****************************************************************************
* * STAAD.Pro CONNECT Edition * *
* Version 22.01.00.** * *
* Proprietary Program of * *
* Bentley Systems, Inc. * *
* Date= APR 14, 2019 * *
* Time= 23:10:18 * *
* Licensed to: Bentley Systems Inc *
*****************************************************************************

1. STAAD SPACE
INPUT FILE: BS5950 2000 - Beam from UB Restrained at Loading.STD
2. START JOB INFORMATION
3. JOB NAME EXAMPLE NO. 4
4. JOB CLIENT THE STEEL CONSTRUCTION INSTITUTE
5. JOB COMMENT SIMPLY SUPPORTED BEAM WITH LATERAL RESTRAINT POINTS
6. ENGINEER DATE JUN-2003
7. END JOB INFORMATION
8. INPUT WIDTH 79
9. UNIT METER KN
10. JOINT COORDINATES
   1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0
12. MEMBER INCIDENCES
   1 1 2; 2 2 3; 3 3 4
14. **************************************************************************
15. START GROUP DEFINITION
16. MEMBER
17.  _MAIN 1 TO 3
18. END GROUP DEFINITION
19. **************************************************************************
20. DEFINE MATERIAL START
21. ISOTROPIC STEEL
22. E 2.05E+08
23. POISSON 0.3
24. DENSITY 76.8195
25. ALPHA 1.2E-05
26. DAMP 0.03
27. END DEFINE MATERIAL
28. MEMBER PROPERTY BRITISH
29. 1 TO 3 TABLE ST UB457X191X82
30. CONSTANTS
31. MATERIAL STEEL ALL
32. SUPPORTS
33. 1 4 FIXED BUT MZ
34. **************************************************************************
35. LOAD 1 DEAD
36. JOINT LOAD
37. 2 FY -40
38. MEMBER LOAD
   STAAD SPACE -- PAGE NO.
39. 1 TO 3 UNI GY -3
40. UNIT MMS KN
41. JOINT LOAD
42. 3 FY -20
43. UNIT METER KN
44. LOAD 2 LIVE
45. JOINT LOAD
46. 2 FY -60
47. UNIT MMS KN
48. JOINT LOAD
49. 3 FY -30
50. LOAD COMB 100 COMBINATION LOAD CASE 3
51. 1 1.4 2 1.6
52. PERFORM ANALYSIS PRINT STATICS CHECK

   PROBLEM STATISTICS
   -----------------------------------
   NUMBER OF JOINTS        4        NUMBER OF MEMBERS        3
   NUMBER OF PLATES        0        NUMBER OF SOLIDS         0
   NUMBER OF SURFACES      0        NUMBER OF SUPPORTS       2
   Using 64-bit analysis engine.
   SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

   TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 14
   TOTAL LOAD COMBINATION CASES = 1 SO FAR.

   STAAD SPACE -- PAGE NO.
### Verification Examples

**V.09 Steel Design**

---

#### STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 1

**DEAD CENTER OF FORCE BASED ON Y FORCES ONLY (MMS).**
*(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)*

<table>
<thead>
<tr>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>0.415517240E+04</td>
</tr>
<tr>
<td>Y</td>
<td>0.000000000E+00</td>
</tr>
<tr>
<td>Z</td>
<td>0.000000000E+00</td>
</tr>
</tbody>
</table>

**TOTAL APPLIED LOAD 1**

#### ***TOTAL APPLIED LOAD (KN MMS) SUMMARY (LOADING 1)**

<table>
<thead>
<tr>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SUMMATION FORCE-X</td>
<td>0.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Y</td>
<td>-87.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Z</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**SUMMATION OF MOMENTS AROUND THE ORIGIN**

<table>
<thead>
<tr>
<th>Moment Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>MX</td>
<td>0.00</td>
</tr>
<tr>
<td>MY</td>
<td>0.00</td>
</tr>
<tr>
<td>MZ</td>
<td>-361500.04</td>
</tr>
</tbody>
</table>

**TOTAL REACTION LOAD 1**

#### ***TOTAL REACTION LOAD (KN MMS) SUMMARY (LOADING 1)**

<table>
<thead>
<tr>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SUMMATION FORCE-X</td>
<td>0.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Y</td>
<td>87.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Z</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**SUMMATION OF MOMENTS AROUND THE ORIGIN**

<table>
<thead>
<tr>
<th>Moment Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>MX</td>
<td>0.00</td>
</tr>
<tr>
<td>MY</td>
<td>0.00</td>
</tr>
<tr>
<td>MZ</td>
<td>361500.04</td>
</tr>
</tbody>
</table>

#### MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 1)

**MAXIMUMS AT NODE**

<table>
<thead>
<tr>
<th>Displacement Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>Y</td>
<td>-1.23549E+00</td>
</tr>
<tr>
<td>Z</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>RX</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>RY</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>RZ</td>
<td>-4.87969E-03</td>
</tr>
</tbody>
</table>

---

#### STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 2

**LIVE CENTER OF FORCE BASED ON Y FORCES ONLY (MMS).**
*(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)*

<table>
<thead>
<tr>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>0.399999997E+04</td>
</tr>
<tr>
<td>Y</td>
<td>0.000000000E+00</td>
</tr>
<tr>
<td>Z</td>
<td>0.000000000E+00</td>
</tr>
</tbody>
</table>

**TOTAL APPLIED LOAD 2**

#### ***TOTAL APPLIED LOAD (KN MMS) SUMMARY (LOADING 2)**

<table>
<thead>
<tr>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SUMMATION FORCE-X</td>
<td>0.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Y</td>
<td>-90.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Z</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**SUMMATION OF MOMENTS AROUND THE ORIGIN**

<table>
<thead>
<tr>
<th>Moment Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>MX</td>
<td>0.00</td>
</tr>
<tr>
<td>MY</td>
<td>0.00</td>
</tr>
<tr>
<td>MZ</td>
<td>-360000.02</td>
</tr>
</tbody>
</table>

**TOTAL REACTION LOAD 2**

#### ***TOTAL REACTION LOAD (KN MMS) SUMMARY (LOADING 2)**

<table>
<thead>
<tr>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SUMMATION FORCE-X</td>
<td>0.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Y</td>
<td>90.00</td>
</tr>
<tr>
<td>SUMMATION FORCE-Z</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**SUMMATION OF MOMENTS AROUND THE ORIGIN**

<table>
<thead>
<tr>
<th>Moment Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>MX</td>
<td>0.00</td>
</tr>
<tr>
<td>MY</td>
<td>0.00</td>
</tr>
<tr>
<td>MZ</td>
<td>360000.02</td>
</tr>
</tbody>
</table>

#### MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 2)

**MAXIMUMS AT NODE**

<table>
<thead>
<tr>
<th>Displacement Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>Y</td>
<td>-1.40263E+00</td>
</tr>
<tr>
<td>Z</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>RX</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>RY</td>
<td>0.00000E+00</td>
</tr>
<tr>
<td>RZ</td>
<td>-4.87969E-03</td>
</tr>
</tbody>
</table>
Verification Examples
V.09 Steel Design

RY = 0.00000E+00 0
RZ = -5.52232E-03 1
************ END OF DATA FROM INTERNAL STORAGE ************
53.  **************************************
54. PARAMETER 1
55. CODE BS5950
56. TRACK 2 ALL
57. MLT _MAIN J1 U2 U3 J4
58. CHECK CODE ALL
STEEL DESIGN

STAAD.Pro CODE CHECKING - (BSI )
***********************
PROGRAM CODE REVISION V2.13_5950-1_2000
STAAD SPACE

5
ALL UNITS ARE - KN   MMS  (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
FX            MY             MZ       LOCATION
=======================================================================
1 ST UB457X191X82  PASS     BS-4.3.6           0.830       100
0.00            0.00      417800.03        0.00
=======================================================================

MATERIAL DATA
Grade of steel           =  S 275
Modulus of elasticity    =  205 kN/mm2
Design Strength (py)    =  275  N/mm2
SECTION PROPERTIES (units - cm)
Member Length =    300.00
Gross Area =  104.00  Net Area =  104.00  Eff. Area =  104.00
z-z  axis       y-y  axis
Moment of inertia        :    37100.004        1870.000
Plastic modulus          :     1830.000         304.000
Effective modulus        :     1830.000         304.000
Shear Area               :       55.094          45.540
DESIGN DATA (units - kN,m)   BS5950-1/2000
Section Class            :   PLASTIC
z-z  axis       y-y  axis
Moment Capacity          :        503.3           80.6
Reduced Moment Capacity  :        503.3           80.6
Shear Capacity           :        909.1          751.4
BUCKLING CALCULATIONS (units - kN,m)
(axis nomenclature as per design code)
LTB Moment Capacity (kNm) and LTB Length (m):   396.39,    3.000
LTB Coefficients & Associated Moments (kNm):
mLT =  0.61 :   mx =  0.00   :  my =  0.00   :  myx =  0.00
Mlt =  417.80 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
BS-4.2.3-(Y)   0.194   100      -     145.6      -       -       -
BS-4.3.6       0.830   100      -     145.6      -    417.8      -
Torsion and deflections have not been considered in the design.
# Material Data

**Grade of steel** = S 275  
**Modulus of elasticity** = 205 kN/mm²  
**Design Strength (py)** = 275 N/mm²

## Section Properties (units - cm)

<table>
<thead>
<tr>
<th>Member Length</th>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gross Area</td>
<td>104.00</td>
<td>104.00</td>
</tr>
<tr>
<td>Moment of inertia</td>
<td>37100.004</td>
<td>1870.00</td>
</tr>
<tr>
<td>Plastic modulus</td>
<td>1830.000</td>
<td>304.00</td>
</tr>
<tr>
<td>Elastic modulus</td>
<td>1613.044</td>
<td>195.504</td>
</tr>
<tr>
<td>Effective modulus</td>
<td>1830.000</td>
<td>304.00</td>
</tr>
<tr>
<td>Shear Area</td>
<td>55.094</td>
<td>45.540</td>
</tr>
</tbody>
</table>

### Design Data (units - kN,m)

**Section Class** = PLASTIC

<table>
<thead>
<tr>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment Capacity</td>
<td>583.3</td>
</tr>
<tr>
<td>Reduced Moment Capacity</td>
<td>583.3 80.6</td>
</tr>
<tr>
<td>Shear Capacity</td>
<td>909.1</td>
</tr>
</tbody>
</table>

## Buckling Calculations (units - kN,m)

LTB Moment Capacity (kNm) and LTB Length (m): 396.39, 3.000

- **mLT = 0.94 : mx = 0.00 : my = 0.00 : myx = 0.00**
- **Mlt = 417.80 : Mx = 0.00 : My = 0.00 : My = 0.00**

## Critical Loads for Each Clause Check (units - kN,m):

<table>
<thead>
<tr>
<th>Clause</th>
<th>Ratio</th>
<th>Load</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>BS-4.2.3-(Y)</td>
<td>0.042</td>
<td>100</td>
<td>-</td>
<td>31.6</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>BS-4.3.6</td>
<td>0.986</td>
<td>100</td>
<td>-</td>
<td>19.0</td>
<td>-</td>
<td>417.8</td>
<td>-</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.

---

STAAD SPACE -- PAGE NO. 7

---

**Verification Examples**

V.09 Steel Design

---

STAAD.Pro 3807 User Manual
V. BS5950 2000 - Beam from UC Restrained at Loading

Design of a simply-supported, 9m beam, which is laterally restrained at the ends and at the points of load application only.

Reference


Problem

The beam is a universal column from S355 steel.

Loads:

- Selfweight \( w_s' = 3.4 \text{kN/m} \)
- Point Load \( W_1' = 122 \text{kN} \)
- Point Load \( W_2' = 61 \text{kN} \)
Table 531: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear capacity, $P_v$ (kN)</td>
<td>465</td>
<td>465.5</td>
<td>none</td>
</tr>
<tr>
<td>Moment capacity, $M_{cx}$ (kN-m)</td>
<td>350</td>
<td>350.5</td>
<td>none</td>
</tr>
<tr>
<td>Lateral-torsional buckling moment, $M_b$ (kN-m)</td>
<td>333</td>
<td>332.17</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

- **SGR 1 ALL**  
  Specifies the use of Grade S355 steel

- **TRACK 2 ALL**  
  Maximum detail output

- **MLT _MAIN J1 U2 U3 J4**  
  Specifies that $m_L$ is to be calculated with the end joints fully restrained and the intermediate joints restraining the upper flange

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Beam from UC Restrained at Loading.STD` is typically installed with the program.
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TO 3 TABLE ST UC254X254X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED BUT MZ
********************************************************************
LOAD 100 FACTORED
JOINT LOAD
2 FY -122
3 FY -61
MEMBER LOAD
1 TO 3 UNI GY -3.4
PERFORM ANALYSIS PRINT STATICS CHECK
********************************************************************
PARAMETER 1
CODE BS5950
SGR 1 ALL
TRACK 2 ALL
MLT _MAIN J1 U2 U3 J4
CHECK CODE ALL
FINISH

STAAD Output
1
********************************************************************
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 23:10:22 *
* Licensed to: Bentley Systems Inc *
********************************************************************
1. STAAD SPACE
INPUT FILE: BS5950 2000 - Beam from UC Restrained at Loading.STD
2. START JOB INFORMATION
3. JOB NAME EXAMPLE NO. 5
4. JOB CLIENT THE STEEL CONSTRUCTION INSTITUTE
5. JOB COMMENT SIMPLY SUPPORTED BEAM WITH LATERAL RERAINT
6. JOB COMMENT AT LOAD APPLICATION POINTS USING A CLASS 3
7. JOB COMMENT UC.
8. ENGINEER DATE JUN-2003
9. END JOB INFORMATION
10. INPUT WIDTH 79
11. UNIT METER KN
12. JOINT COORDINATES
13. 1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0
14. MEMBER INCIDENCES
15. 1 1 2; 2 2 3; 3 3 4
16. ***********************************************
17. START GROUP DEFINITION
18. MEMBER
19. MAIN 1 TO 3
20. END GROUP DEFINITION
21. ***********************************************
22. DEFINE MATERIAL START
23. ISOTROPIC STEEL
24. E 2.05E+08
25. POISSON 0.3
26. DENSITY 76.8195
27. ALPHA 1.2E-05
28. DAMP 0.03
29. END DEFINE MATERIAL
30. MEMBER PROPERTY BRITISH 1 TO 3 TABLE ST UC254X254X73
31. CONSTANTS
32. MATERIAL STEEL ALL
33. SUPPORTS
34. 1 4 FIXED BUT MZ
35. ***********************************************
36. LOAD 100 FACTORED
37. JOINT LOAD
38. STAAD SPACE -- PAGE NO.

39. 2 FY -122
40. 3 FY -61
41. MEMBER LOAD
42. 1 TO 3 UNI GY -3.4
43. PERFORM ANALYSIS PRINT STATICS CHECK

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.   100 FACTORED
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)

X = 0.407162924E+01
Y = 0.000000000E+00
Z = 0.000000000E+00

TOTAL APPLIED LOAD   100
***TOTAL APPLIED LOAD ( KN METE ) SUMMARY (LOADING 100 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -213.60
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-MX= 0.00 MY= 0.00 MZ= -869.70

TOTAL REACTION LOAD 100
***TOTAL REACTION LOAD( KN METE ) SUMMARY (LOADING 100 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 213.60
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.00 MY= 0.00 MZ= 869.70
MAXIMUM DISPLACEMENTS (CM /RADIANS) (LOADING 100)
MAXIMUMS AT NODE
X = 0.0000E+00 0
Y = -1.0280E+01 2
Z = 0.0000E+00 0
RX= 0.0000E+00 0
RY= 0.0000E+00 0
RZ= -4.0961E-02 1
************ END OF DATA FROM INTERNAL STORAGE ************
44. ****************************************
45. PARAMETER 1
46. CODE BS5950
47. SGR 1 ALL
48. TRACK 2 ALL
49. MLT _MAIN J1 U2 U3 J4
50. CHECK CODE ALL

STEEL DESIGN
STAAD SPACE -- PAGE NO. 4

STAAD.Pro CODE CHECKING - (BSI )
***********************
PROGRAM CODE REVISION V2.13_5950-1_2000
STAAD SPACE -- PAGE NO. 5

ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
FX            MY             MZ       LOCATION
=======================================================================
1 ST UC254X254X73  PASS     BS-4.3.6           0.958       100
0.00            0.00         335.60        0.00
=======================================================================

MATERIAL DATA
Grade of steel = S 355
Modulus of elasticity = 205 kN/mm2
Design Strength (py) = 355 N/mm2
SECTION PROPERTIES (units - cm)
Member Length = 300.00
Gross Area = 93.10 Net Area = 93.10 Eff. Area = 93.10
z-z axis       y-y axis
Moment of inertia : 11400.002        3910.000
Plastic modulus : 992.000          465.000
Elastic modulus : 897.285          307.148
Effective modulus : 986.822         456.370
Shear Area : 65.076               21.853
DESIGN DATA (units - kN,m) BS5950-1/2000
Section Class : SEMI-COMPACT
z-z axis       y-y axis
Moment Capacity : 350.3             162.0
Reduced Moment Capacity : 350.3        162.0
Shear Capacity : 1386.1             465.5

BUCKLING CALCULATIONS (units - kN,m)
(axis nomenclature as per design code)
LTB Moment Capacity (kNm) and LTB Length (m): 332.21, 3.000
LTB Coefficients & Associated Moments (kNm):
mLT = 0.61 : mx = 0.00 : my = 0.00 : myx = 0.00
Mlt = 335.60 : Mx = 0.00 : My = 0.00 : My = 0.00

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
CLAUSE RATIO LOAD FX VY VZ MZ MY
BS-4.2.3-(Y) 0.251 100 - 117.0 - - -
BS-4.3.6 0.958 100 - 117.0 - 335.6 -

Torsion and deflections have not been considered in the design.

STAAD SPACE

ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
2 ST UC254X254X73 PASS BS-4.3.6 0.958 100
0.00 0.00 335.60 0.00
=======================================================================

MATERIAL DATA
Grade of steel = S 355
Modulus of elasticity = 205 kN/mm²
Design Strength (py) = 355 N/mm²

SECTION PROPERTIES (units - cm)
Member Length = 300.00
Gross Area = 93.10 Net Area = 93.10 Eff. Area = 93.10
z-z axis y-y axis
Moment of inertia : 11400.002 3910.000
Plastic modulus : 992.000 465.000
Elastic modulus : 897.285 307.148
Effective modulus : 986.822 456.370
Shear Area : 65.076 21.853

DESIGN DATA (units - kN,m) BS5950-1/2000
Section Class : SEMI-COMPACT
z-z axis y-y axis
Moment Capacity : 350.3 162.0
Reduced Moment Capacity : 350.3 162.0
Shear Capacity : 1386.1 465.5

BUCKLING CALCULATIONS (units - kN,m)
(axis nomenclature as per design code)
LTB Moment Capacity (kNm) and LTB Length (m): 332.21, 3.000
LTB Coefficients & Associated Moments (kNm):
mLT = 0.94 : mx = 0.00 : my = 0.00 : myx = 0.00
Mlt = 335.60 : Mx = 0.00 : My = 0.00 : My = 0.00

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
CLAUSE RATIO LOAD FX VY VZ MZ MY
BS-4.2.3-(Y) 0.055 100 - 25.4 - - -
BS-4.3.6 0.784 100 - 15.2 - 335.6 -

Torsion and deflections have not been considered in the design.

STAAD SPACE

ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
3 ST UC254X254X73 PASS BS-4.3.6 0.784 100
0.00 0.00 274.60 0.00
=======================================================================

MATERIAL DATA
V. BS5950 2000 - Pinned Column Using Non-Slender UC

A 6.0 m column is pin ended about both axes and has no intermediate restraints.
Reference


Problem

The column is designed in S275 steel for the factored loading.

Point load, \( F_c = 2,500 \text{ kN} \)

Comparison

Table 532: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness about strong axis, ( \lambda_x )</td>
<td>38.5</td>
<td>38.299</td>
<td>none</td>
</tr>
<tr>
<td>Slenderness about weak axis, ( \lambda_y )</td>
<td>63.6</td>
<td>63.569</td>
<td>none</td>
</tr>
<tr>
<td>Compression capacity, ( P_c ) (kN)</td>
<td>3,100</td>
<td>3,090</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

```
TRACK 2 ALL
SGR 0 ALL
```

Maximum detail output

Identifies steel grade as S275

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Pinned Column Using Non-Slender UC.STD is typically installed with the program.

STAAD SPACE

```
START JOB INFORMATION
JOB NAME Example no. 10
JOB CLIENT The Steel Construction Institute
JOB COMMENT Pinned column using a non-slender UC.
ENGINEER DATE Jun-2003
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT METER KN
```

```
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
```
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TABLE ST UC356X368X129
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
******************************************************************************
LOAD 1 AXIAL LOAD
JOINT LOAD
2 FY -2500
PERFORM ANALYSIS PRINT STATICS CHECK
******************************************************************************
PARAMETER 1
CODE BS5950
SGR 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

1
******************************************************************************
*
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 23:10:32 *
*
* Licensed to: Bentley Systems Inc *
******************************************************************************
1. STAAD SPACE
INPUT FILE: BS5950 2000 - Pinned Column Using Non-Slender UC.STD
2. START JOB INFORMATION
3. JOB NAME EXAMPLE NO. 10
4. JOB CLIENT THE STEEL CONSTRUCTION INSTITUTE
5. JOB COMMENT PINNED COLUMN USING A NON-SLENDER UC.
6. ENGINEER DATE JUN-2003
7. END JOB INFORMATION
8. INPUT WIDTH 79
9. UNIT METER KN
10. JOINT COORDINATES
11. 1 0 0 0; 2 0 6 0
12. MEMBER INCIDENCES
13. 1 1 2
14.****************************************************************************
15. DEFINE MATERIAL START
16. ISOTROPIC STEEL
17. E 2.05E+08
18. POISSON 0.3
19. DENSITY 76.8195
20. ALPHA 1.2E-05
21. DAMP 0.03
22. END DEFINE MATERIAL
23. MEMBER PROPERTY BRITISH
24. 1 TABLE ST UC356X368X129
25. CONSTANTS
26. MATERIAL STEEL ALL
27. SUPPORTS
28. 1 FIXED
29. *************************************
30. LOAD 1 AXIAL LOAD
31. JOINT LOAD
32. 2 FY -2500
33. PERFORM ANALYSIS PRINT STATICS CHECK

--- PAGE NO. 2 ---

PROBLEM STATISTICS
---------------------
NUMBER OF JOINTS 2
NUMBER OF MEMBERS 1
NUMBER OF PLATES 0
NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0
NUMBER OF SUPPORTS 1
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 6
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

--- PAGE NO. 3 ---

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 1
AXIAL LOAD
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.000000000E+00
Y = 0.599999997E+01
Z = 0.000000000E+00

TOTAL APPLIED LOAD 1
***TOTAL APPLIED LOAD ( KN METE ) SUMMARY (LOADING 1 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -2500.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.0000000E+00 MY= 0.0000000E+00 MZ= 0.0000000E+00

TOTAL REACTION LOAD 1
***TOTAL REACTION LOAD( KN METE ) SUMMARY (LOADING 1 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 2500.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.0000000E+00 MY= 0.0000000E+00 MZ= 0.0000000E+00

MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING 1)
MAXIMUMS AT NODE
X = 0.00000E+00 0
Y = -4.46163E-01 2
Z = 0.00000E+00 0
RX= 0.00000E+00 0
RY= 0.00000E+00 0
RZ= 0.00000E+00 0
MATERIAL DATA
Grade of steel = S 275
Modulus of elasticity = 205 kN/mm²
Design Strength (py) = 265 N/mm²

SECTION PROPERTIES (units - cm)
Member Length = 600.00
Gross Area = 164.00 Net Area = 164.00 Eff. Area = 164.00

z-z axis y-y axis
Moment of inertia : 40200.008 14600.002
Plastic modulus : 2480.000 1200.000
Elastic modulus : 2260.968 792.187
Effective modulus : 2458.506 1159.980
Shear Area : 116.109 36.982

DESIGN DATA (units - kN,m) BS5950-1/2000
Section Class = SEMI-COMPACT
Squash Load : 4346.00
Axial force/Squash load : 0.575
z-z axis y-y axis
Compression Capacity : 3992.2 3089.3
Shear Capacity : 1846.1 588.0

BUCKLING CALCULATIONS (units - kN,m)
(axis nomenclature as per design code)

Critical Loads for each Clause Check (units- kN,m):
Clause Ratio Load FX VY VZ MZ MY
BS-4.7 (C) 0.809 1 2500.0 - - - -

Torsion and deflections have not been considered in the design.
V. BS5950 2000 - Pinned Column Using Non-Slender RHS

A 6.0m rectangular hollow section (RHS) column is pin ended about both axes and has no intermediate restraints.

Reference


Problem

The column is designed in S355 steel for the factored loading.

Point load, $F_c = 2,500 \text{ kN}$

Comparison

Table 533: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness, $\lambda$</td>
<td>61.4</td>
<td>61.424</td>
<td>none</td>
</tr>
<tr>
<td>Compression capacity, $P_c$ (kN-m)</td>
<td>2,790</td>
<td>2,794.6</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

<table>
<thead>
<tr>
<th>TRACK 2 ALL</th>
<th>Maximum detail output</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGR 1 ALL</td>
<td>Identifies steel grade as S355</td>
</tr>
</tbody>
</table>

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.
The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Pinned Column Using Non-Slender RHS.STD` is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
JOB NAME Example no. 11
JOB CLIENT The Steel Construction Institute
JOB COMMENT Pinned column using a non-slender RHS.
ENGINEER DATE Jun 2003
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TABLE ST TUB25025010.0
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 AXIAL LOAD
JOINT LOAD
2 FY -2500
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER 1
CODE BS5950
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
* Licensed to: Bentley Systems Inc *
***************************************
1. STAAD SPACE
INPUT FILE: BS5950 2000 - Pinned Column Using Non-Slender RHS.STD
2. START JOB INFORMATION
3. JOB NAME EXAMPLE NO. 11
4. JOB CLIENT THE STEEL CONSTRUCTION INSTITUTE
5. JOB COMMENT PINNED COLUMN USING A NON-SLENDER RHS.
6. ENGINEER DATE JUN 2003
7. END JOB INFORMATION
8. INPUT WIDTH 79
9. UNIT METER KN
10. JOINT COORDINATES
11. 1 0 0 0; 2 0 6 0
12. MEMBER INCIDENTS
13. 1 1 2
14. *************************************
15. DEFINE MATERIAL START
16. ISOTROPIC STEEL
17. E 2.05E+08
18. POISSON 0.3
19. DENSITY 76.8195
20. ALPHA 1.2E-05
21. DAMP 0.03
22. END DEFINE MATERIAL
23. MEMBER PROPERTY BRITISH
24. 1 TABLE ST TUB25025010.0
25. CONSTANTS
26. MATERIAL STEEL ALL
27. SUPPORTS
28. 1 FIXED
29. *************************************
30. LOAD 1 AXIAL LOAD
31. JOINT LOAD
32. 2 FY -2500
33. PERFORM ANALYSIS PRINT STATICS CHECK
STAAD SPACE

Problem Statistics
-----------------------------------
NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
NUMBER OF PLATES          0  NUMBER OF SOLIDS       0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     1
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       6
TOTAL LOAD COMBINATION CASES =     0 SO FAR.

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
AXIAL LOAD
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.000000000E+00
Y =  0.599999997E+01
Z =  0.000000000E+00
TOTAL APPLIED LOAD     1
**TOTAL APPLIED LOAD (KN METE) SUMMARY (LOADING 1)**

SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -2500.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.0000000E+00  MY= 0.0000000E+00  MZ= 0.0000000E+00

**TOTAL REACTION LOAD 1**

***TOTAL REACTION LOAD (KN METE) SUMMARY (LOADING 1)**

SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 2500.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.0000000E+00  MY= 0.0000000E+00  MZ= 0.0000000E+00

MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 1)

MAXIMUMS AT NODE
X = 0.00000E+00 0
Y = -7.71030E-01 2
Z = 0.00000E+00 0
RX= 0.00000E+00 0
RY= 0.00000E+00 0
RZ= 0.00000E+00 0

*********** END OF DATA FROM INTERNAL STORAGE ***********

34. ***************************************
35. PARAMETER 1
36. CODE BS5950
37. SGR 1 ALL
38. TRACK 2 ALL
39. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE -- PAGE NO.

4

STAAD.Pro CODE CHECKING - (BSI)

PROGRAM CODE REVISION V2.13_5950-1_2000

STAAD SPACE -- PAGE NO.

5

ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)

MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION

===============================================================
1 ST TUB25025010.0 PASS BS-4.7 (C) 0.895 1
2500.00 C 0.00 0.00 0.00

===============================================================

MATERIAL DATA

Grade of steel = S 355
Modulus of elasticity = 205 kN/mm2
Design Strength (py) = 355 N/mm2

SECTION PROPERTIES (units - cm)

Member Length = 600.00
Gross Area = 94.90 Net Area = 94.90 Eff. Area = 94.90

z-z axis y-y axis

Moment of inertia : 9055.001 9055.001
Plastic modulus : 851.000 851.000
Elastic modulus : 724.400 724.400
Effective modulus : 851.000 851.000
Shear Area : 47.450 47.450

DESIGN DATA (units - kN,m) BS5950-1/2000

Section Class : SEMI-COMPACT
V. BS5950 2000 - Pinned Column Using Slender CHS

A 6.0m circular hollow section (CHS) column is pin ended about both axes and has no intermediate restraints.

Reference


Problem

The column is designed in S355 steel for the factored loading.

Point load, $F_c = 2,500 \text{ kN}$
Comparison

Table 534: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Effective area, $A_{\text{eff}}$ (cm$^2$)</td>
<td>77.6</td>
<td>77.63</td>
<td>none</td>
</tr>
<tr>
<td>Slenderness, $\lambda$</td>
<td>42.2</td>
<td>42.413</td>
<td>none</td>
</tr>
<tr>
<td>Compression capacity, $P_c$ (kN-m)</td>
<td>2,561</td>
<td>2,562.3</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

```
TRACK 2 ALL
Maximum detail output
SGR 1 ALL
Identifies steel grade as S355

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.
```

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000\BS5950 2000 - Pinned Column Using Slender CHS.STD is typically installed with the program.
2 FY -2500
PERFORM ANALYSIS PRINT STATICS CHECK
******************************************************
PARAMETER 1
CODE BS5950
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

*************** STAAD Output *********************

1

******************************************************************************************************************
*                                                                                                           *
*           STAAD.Pro CONNECT Edition                     *                                             *
*           Version  22.01.00.**                             *                                             *
*           Proprietary Program of                           *                                             *
*           Bentley Systems, Inc.                            *                                             *
*           Date= APR 14, 2019                               *                                             *
*           Time= 23:10:13                                    *                                             *
*                                                                                                           *
*           Licensed to: Bentley Systems Inc                   *                                             *
******************************************************************************************************************

1. STAAD SPACE
INPUT FILE: BS5950 2000 - Pinned Column Using Slender CHS.STD
2. START JOB INFORMATION
3. JOB NAME EXAMPLE NO. 12
4. JOB CLIENT THE STEEL CONSTRUCTION INSTITUTE
5. JOB COMMENT PINNED COLUMN USING A SLENDER CHS.
6. ENGINEER DATE JUN-2003
7. END JOB INFORMATION
8. INPUT WIDTH 79
9. UNIT METER KN
10. JOINT COORDINATES
11. 1 0 0 0; 2 0 6 0
12. MEMBER INCIDENCES
13. 1 1 2
14. ****************************************
15. DEFINE MATERIAL START
16. ISOTROPIC STEEL
17. E 2.05E+08
18. POISSON 0.3
19. DENSITY 76.8195
20. ALPHA 1.2E-05
21. DAMP 0.03
22. END DEFINE MATERIAL
23. MEMBER PROPERTY BRITISH
24. 1 TABLE ST PIP4066.3
25. CONSTANTS
26. MATERIAL STEEL ALL
27. SUPPORTS
28. 1 FIXED
29. ****************************************
30. LOAD 1 AXIAL FORCE
31. JOINT LOAD
32. **FY -2500**
33. PERFORM ANALYSIS PRINT STATICS CHECK
   STAAD SPACE -- PAGE NO.

---

2

**PROBLEM STATISTICS**

-----------------------------------

NUMBER OF JOINTS 2  NUMBER OF MEMBERS 1
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 1

Using 64-bit analysis engine.

SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 6
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

---

3

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 1

AXIAL FORCE

CENTER OF FORCE BASED ON Y FORCES ONLY (METE).

(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)

X = 0.0000000000E+00
Y = 0.599999997E+01
Z = 0.0000000000E+00

TOTAL APPLIED LOAD 1

***TOTAL APPLIED LOAD ( KN METE ) SUMMARY (LOADING 1 )

SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -2500.00
SUMMATION FORCE-Z = 0.00

SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.0000000E+00  MY= 0.0000000E+00  MZ= 0.0000000E+00

TOTAL REACTION LOAD 1

***TOTAL REACTION LOAD(O KN METE ) SUMMARY (LOADING 1 )

SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 2500.00
SUMMATION FORCE-Z = 0.00

SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.0000000E+00  MY= 0.0000000E+00  MZ= 0.0000000E+00

MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING 1)

MAXIMUMS AT NODE

X = 0.00000E+00  0
Y = -9.23873E-01  2
Z = 0.00000E+00  0
RX= 0.00000E+00  0
RY= 0.00000E+00  0
RZ= 0.00000E+00  0

********** END OF DATA FROM INTERNAL STORAGE **********

34. **************************************
35. PARAMETER 1
36. CODE BS5950
37. SGR 1 ALL
38. TRACK 2 ALL
39. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE -- PAGE NO.

---

4

STAAD.Pro CODE CHECKING - (BSI )

**************************************************

PROGRAM CODE REVISION V2.13_5950-1_2000

STAAD SPACE -- PAGE NO.

---
ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)

<table>
<thead>
<tr>
<th>MEMBER TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>Location</td>
</tr>
<tr>
<td>1 ST PIP4066.3</td>
<td>PASS</td>
<td>BS-4.7 (C)</td>
<td>0.976</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>2500.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

MATERIAL DATA
Grade of steel = S 355
Modulus of elasticity = 205 kN/mm²
Design Strength (py) = 355 N/mm²

SECTION PROPERTIES (units - cm)
Member Length = 600.00
Gross Area = 79.20 Net Area = 79.20 Eff. Area = 77.63

<table>
<thead>
<tr>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment of inertia : 15850.002</td>
<td>15850.002</td>
</tr>
<tr>
<td>Plastic modulus : 1009.000</td>
<td>1009.000</td>
</tr>
<tr>
<td>Elastic modulus : 780.020</td>
<td>780.020</td>
</tr>
<tr>
<td>Effective modulus : 780.020</td>
<td>780.020</td>
</tr>
<tr>
<td>Shear Area : 47.520</td>
<td>47.520</td>
</tr>
</tbody>
</table>

DESIGN DATA (units - kN,m) BS5950-1/2000
Section Class : SLENDER
Squash Load : 2811.60
Axial force/Squash load : 0.889

<table>
<thead>
<tr>
<th>z-z axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression Capacity : 2562.3</td>
<td>2562.3</td>
</tr>
<tr>
<td>Shear Capacity : 1012.2</td>
<td>1012.2</td>
</tr>
</tbody>
</table>

BUCKLING CALCULATIONS (units - kN,m)
(axis nomenclature as per design code)

<table>
<thead>
<tr>
<th>x-x axis</th>
<th>y-y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness : 42.413</td>
<td>42.413</td>
</tr>
<tr>
<td>Radius of gyration (cm) : 14.147</td>
<td>14.147</td>
</tr>
<tr>
<td>Effective Length : 6.000</td>
<td>6.000</td>
</tr>
</tbody>
</table>

LTB check unnecessary for this section

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>BS-4.7 (C)</td>
<td>0.976</td>
<td>1</td>
<td>2500.0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Torsion and deflections have not been considered in the design.

--- PAGE NO.

************** END OF TABULATED RESULT OF DESIGN **************

FINISH

***** END OF THE STAAD.Pro RUN *******
**** DATE= APR 14,2019 TIME= 23:10:14 ****

For technical assistance on STAAD.Pro, please visit
Details about additional assistance from
Bentley and Partners can be found at program menu
Help->Technical Support
Copyright (c) 1997-2017 Bentley Systems, Inc.
http://www.bentley.com
V. United States

V. AASHTO

V. AASHTO 17th Ed ASD - Design Frame
The following compares the solution of a design performed using STAAD.Pro against a hand calculation.

Reference

Related Links
- D1.D.1 AASHTO (ASD) (on page 1464)

Problem
Determine the allowable stresses (AASHTO code) for the members of the structure as shown in figure. Also, perform a code check for these members based on the results of the analysis.

Members 1, 2 = W12X26, Members 3, 4 = W14X43
Members 5, 6, 7 = W16X36, Member 8 = L40404,
Member 9 = L50506

The frame is subject to the following load cases:

1. a uniform gravity load along the beam, \( w = 2 \text{ kips/ft} \) (dead + live)
2. a lateral wind load, \( F = 15 \text{ kips} \)
3. a combination of 75\% of load 1 and 75\% of load 2

**Solution**

Only the AASHTO steel design elements are check here. No structural analysis calculations are included in these hand verifications.

Though the program does check shear per the AASHTO specifications, those calculations are not reflected here. Only the controlling stress ratios are presented.

As all members are grade 36 steel, the following critical slenderness parameter applies to each:

\[
C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 \times 29,000}{36}} = 126.1
\]

**Member 1**

Size W 12X26, \( L = 10 \text{ ft} \), \( a = 7.65 \text{ in} \), \( S_z = 33.39 \text{ in}^3 \)

From observation, Load case 1 will govern

- \( F_x = 25.0 \text{ kip (compression)} \)
- \( M_z = 56.5 \text{ k-ft} \)

Calculate the allowable stress as per Table 10.32.1A.

**Bending Minor Axis**

Allowable minor axis bending stress:

\[
FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}
\]

**Bending Major Axis**

\[
F_{cz} = \frac{50(10)^6C_b}{S_{xc}} \left( I_{yc} \right) \left( 0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{1} \right)^2 \right) \leq 0.55F_y
\]

Where:

\[
C_b = 1.75 + 1.05(M1/M2)+0.3x(M1/M2)^2
\]

\( M1 = 0 \), so \( C_b = 1.75 \)

\( S_{xc} = \) Section modulus with respect to the compression flange \( = \frac{204}{0.5 \cdot 12.22} = 33.38789 \text{ in}^3 \)

\[
I_{YC} = t_b^3/12 = 0.38 \cdot 6.49^3/12 = 8.6564 \text{ in}^4
\]

\[
J = (2 \times 6.49 \cdot 0.38^3+ (12.22 – 2 \cdot 0.38) \cdot 0.23^3)/3 = 0.28389 \text{ in}^4
\]

\[
F_{cz} = \frac{50(10)^61.75}{33.38789} \left( \frac{8.6564}{120} \right) \left( 0.772 \frac{0.28389}{8.6564} + 9.87 \left( \frac{11.46}{120} \right)^2 \right) \leq 0.55 \cdot 19.8 = 64.375 \text{ ksi}
\]

which is larger than \( 0.55 \cdot F_Y = 19.8 \text{ ksi} \), so \( F_{cz} = 19.8 \text{ ksi} \)

**Axial Compression**

Critical \( (kL/r) = 1.0 \cdot 120/1.5038 = 79.7978 \)

As \( (kL/r) < C_c \), the allow axial stress in compression is given by:
Actual Stress

Actual axial stress, \( f_a = \frac{25}{7.65} = 3.26 \text{ ksi} \)

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10.36 to calculate the design ratio.

Actual bending stress = \( f_{bz} = 56.5 \cdot 12 / 33.4 = 1.692 \cdot 12 = 20.3 \text{ ksi} \)

\[
F_{ez} = \frac{n^2E}{F.S.(kL/r)^2} = \frac{n^229,000}{2.12(120/5.17)^2} = 250.6 \text{ksi}
\]

From Table 10-36A, \( C_{mz} = 0.85 \)

Equation 10-42

\[
\frac{f_a}{F_a} + \frac{C_{mz}f_{bz}}{F_{bz}} + \frac{C_{my}f_{by}}{F_{by}} = \frac{3.26}{13.58} + \frac{0.58 \cdot 20.3}{19.8} + 0 = 1.122
\]

For the end section, use Equation 10.43:

\[
\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{3.26}{0.472(36)} + \frac{20.3}{19.8} + 0 = 1.217
\]

The critical stress ratio is thus 1.217. The value calculated by STAAD is 1.218

**Member 2**

Size W 12X26, L = 5 ft., \( a = 7.65 \text{ in}^2, Sz = 33.4 \text{ in}^3 \)

From observation Load case 1 will govern, Forces at the midspan are

- \( F_x = 8.71 \text{ kip (compression)} \)
- \( M_z = 56.5 \text{ k-ft} \)

Calculate the allowable stress as per Table 10.32.1A.

**Bending Minor Axis**

Allowable minor axis bending stress:

\( F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi} \)

**Bending Major Axis**

\[
F_{cz} = \frac{50(10)^6C_b}{S_{xc}} \left( \frac{I_{xc}}{I_{yc}} \right) \left( \frac{0.772 \frac{f}{f_{yc}} + 9.87 \frac{d}{I_{yc}})^2}{0.55F_y} \right) \leq 0.55F_y
\]

Where:

\( C_b = 1.75 + 1.05(M1/M2)+0.3x(M1/M2)^2 \)

\( M1 = 39.44 \text{ and } M2 = 677.96, \text{ so } C_b = 1.69 \)

\( S_{xc} = \text{Section modulus with respect to the compression flange} = 204/(0.5 \cdot 12.22) = 33.38789 \text{ in}^3 \)
\[ I_{YC} = t^3/12 = 0.38 \cdot 6.49^3/12 = 8.6564 \text{ in}^4 \]
\[ J = (2 \cdot 6.49 \cdot 0.38)^3 + (12.22 - 2 \cdot 0.38) \cdot 0.23^3/3 = 0.28389 \text{ in}^4 \]
\[ F_{ez} = \frac{50(10)^{6/3}}{33.33739} \left( \frac{8.6564}{60} \right) \left( 0.772 \cdot \frac{0.28389}{8.6564} + 9.87 \left( \frac{11.46}{60} \right)^2 \right)^{10^{-3}} = 227.34 \text{ ksi} \]

which is larger than \( 0.55 \cdot F_Y = 19.8 \text{ ksi} \), so \( F_{CZ} = 19.8 \text{ ksi} \)

**Axial Compression**

Critical \( (kL/r) = 1.0 \cdot 60/1.504 = 39.92 \)

As \( (kL/r) < C_c \), the allow axial stress in compression is given by:

\[ F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(kL/r)^2 F_y}{4\pi^2 E} \right] \]
\[ = \frac{36}{2.12} \left[ 1 - \frac{(39.92)^2 36}{4\pi^2 29,000} \right] = 16.13 \text{ ksi} \]

**Actual Stress**

Actual axial stress, \( f_a = 8.71 / 7.65 = 1.138 \text{ ksi} \)

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = \( f_{bz} = 56.5 \cdot 12/33.4 = 1.691 \cdot 12 = 20.3 \text{ ksi} \)

\( (KL/r)_z = 1 \cdot 60/5.16 = 11.618 \)

\[ F_{ez} = \frac{x^2 E}{F.S.(kL/r)^2} \]
\[ = \frac{229,000}{2.12(11.618)^2} = 998.5 \text{ ksi} \]

From Table 10-36A, \( C_{mz} = 0.85 \)

Equation 10-42

\[ \frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{F_{bz}} + \frac{C_{my} f_{by}}{F_{by}} = \frac{1.138}{16.13} + \frac{0.58 \cdot 20.3}{19.8} + 0 = 0.942 \]

For the end section, use Equation 10.43:

\[ \frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{1.138}{0.472(36)} + \frac{20.3}{19.8} + 0 = 1.092 \]

The critical stress ratio is thus 1.092. The value calculated by STAAD is 1.093.

**Member 3**

Size W 14X43, L = 11 ft., \( a = 12.6 \text{ in}^2 \), \( Sz = 62.7 \text{ in}^3 \)

From observation Load case 3 will govern, Forces at the end are

- \( F_x = 25.5 \text{ kip (compression)} \)
- \( M_x = 112.17 \text{ k-ft} \)

Calculate the allowable stress as per Table 10.32.1A.

**Bending Minor Axis**

Allowable minor axis bending stress:
FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}

**Bending Major Axis**

\[ F_{cz} = \frac{50(10)^6 C_b}{S_{xc}} \left( \frac{I_{yc}}{I_{yc}^2} \right) \left[ 0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{r} \right)^2 \right] \leq 0.55 F_Y \]

Where:

\[ C_b = 1.75 + 1.05(M_1/M_2) + 0.3x(M_1/M_2)^2 \]

\[ M_1 = 0, \text{ so } C_b = 1.75 \]

\[ S_{zc} = \text{Section modulus with respect to the compression flange} = \frac{428}{0.5 \cdot 13.66} = 62.7 \text{ in}^3 \]

\[ I_{YC} = \frac{tb}{12} = 0.53 \cdot 8.0 \frac{3}{12} = 22.61 \text{ in}^4 \]

\[ J = \frac{2 \cdot 8.0 \cdot 0.53 + (13.66 - 2 \cdot 0.53) \cdot 0.305}{3} = 0.917 \text{ in}^4 \]

\[ F_{cz} = \frac{50(10)^6 1.75}{62.7} \left( \frac{22.61}{12.6} \right) \left[ 0.772 \frac{0.917}{22.61} + 9.87 \left( \frac{12.6}{132} \right)^2 \right] 10^{-3} = 83.19 \text{ ksi} \]

which is larger than \( 0.55 \cdot F_Y = 19.8 \text{ ksi} \), so \( F_{cz} = 19.8 \text{ ksi} \)

**Axial Compression**

Critical \( (kL/r) = 1.0 \cdot 132/1.894 = 69.69 \)

As \( (kL/r) < C_c \), the allow axial stress in compression is given by:

\[ F_a = \frac{F_y}{F.S.} \left( 1 - \frac{(kL/r)^2 F_y}{\pi^2 E} \right) = \frac{36}{2.12} \left( 1 - \frac{(69.69)^2 36}{\pi^2 29,000} \right) = 14.39 \text{ ksi} \]

**Actual Stress**

Actual axial stress, \( f_a = 25.5 / 12.6 = 2.024 \text{ ksi} \)

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress \( f_{bz} = 112.17 \cdot 12/62.7 = 1.789 \cdot 12 = 21.467 \text{ ksi} \)

\( (KL/r)_e = 1 \cdot 132/5.828 = 22.649 \)

\[ F_{ez} = \frac{n^2 E}{F.S.(kL/r)^2} = \frac{n^2 29,000}{2.12(22.648)^2} = 263.18 \text{ ksi} \]

From Table 10-36A, \( C_{mz} = 0.85 \)

Equation 10-42

\[ \frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{F_{bz}} + \frac{C_{my} f_{by}}{F_{by}} = \frac{2.024}{14.39} \frac{0.58 \cdot 21.467}{263.21^{19.8}} + 0 = 1.069 \]

For the end section, use Equation 10.43:

\[ \frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{2.024}{0.472(36)} + \frac{21.467}{19.8} + 0 = 1.203 \]

The critical stress ratio is thus 1.203. The value calculated by STAAD is 1.204.
Member 4
Size W 14X43, L = 4 ft., a = 12.6 in², Sz = 62.6 in³
From observation Load case 3 will govern, Forces at the end are
- \( F_x = 8.75 \text{ kip (tension)} \)
- \( M_z = 112.17 \text{ k-ft} \)
Calculate the allowable stress as per Table 10.32.1A.

**Bending Minor Axis**
Allowable minor axis bending stress:
\[
FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}
\]

**Bending Major Axis**
\[
F_{cz} = \frac{50(10^6)C_b}{S_{xc}} \left( \frac{I_{xc}}{I} \right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{I} \right)^2} \leq 0.55F_y
\]
Where:
\[
C_b = 1.75 + 1.05(M1/M2)+0.3 \cdot (M1/M2)^2
\]
\[
M1 = -191.36 \text{ Kip-in }, M2 = -1346.08 \text{ Kip-in so } C_b = 1.606
\]
\[
I_{YC} = \frac{tb^3}{12} = 0.53 \cdot \frac{8.0^3}{12} = 22.61 \text{ in}^4
\]
\[
J = (2 \cdot 8.0 \cdot 0.53^3 + (13.66 - 2 \cdot 0.53) \cdot 0.305^3)/3 = 0.913 \text{ in}^4
\]
\[
F_{cz} = \frac{50(10^6)1.606}{62.6} \left( \frac{22.61}{48} \right) \sqrt{0.772 \frac{0.911}{22.61} + 9.87 \left( \frac{12.6}{48} \right)^2} \cdot 10^{-3} = 508.8 \text{ ksi}
\]
which is larger than \( 0.55 \cdot F_Y = 19.8 \text{ ksi} \), so \( F_{cz} = 19.8 \text{ ksi} \)

**Axial Tension**

**Note:** No connection information is specified and no reduction of section is assumed.

\[
F_t = 0.55 \cdot F_Y = 19.8 \text{ ksi}
\]

**Actual Stress**
Actual axial stress, \( f_a = 8.75 / 12.6 = 0.694 \text{ ksi} \)
Actual bending stress = \( f_{bz} = 112.17 \cdot 12 / 62.7 = 1.789 \cdot 12 = 21.47 \text{ ksi} \), which exceeds \( F_{cz} \).
\[
f_{bz}/F_{cz} = 21.47/19.8 = 1.084
\]
The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.43 in section 10-36 to calculate the design ratio for the end section.
\[
\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.694}{0.472(36)} + \frac{21.47}{19.8} + 0 = 1.125
\]
The critical stress ratio is thus 1.125. The value calculated by STAAD is 1.126.

Member 5
Size W 16X36, L = 5 ft., a = 10.6 in², Sz = 56.5 in³
From observation Load case 3 will govern, Forces at the end are

- \( F_x = 14.02 \) kip (compression)
- \( M_z = 57.04 \) k-ft

Calculate the allowable stress as per Table 10.32.1A.

### Bending Minor Axis

Allowable minor axis bending stress:

\[ F_{ty} = F_{tz} = 0.55 \cdot F_Y = 19.8 \text{ ksi} \]

### Bending Major Axis

\[
F_{ez} = \frac{50(10)^6 C_b}{S_{xc}} \left( \frac{I_{yc}}{I} \right) \left[ 0.772 \frac{J}{I_{yc}} + 9.87 \left( \frac{d}{r} \right)^2 \right] \leq 0.55 F_Y
\]

Where:

\[ C_b = 1.75 + 1.05(M1/M2)+0.3 \cdot (M1/M2)^2 \]

\[ M1 = 40.14, M2 = -684.4 \text{ so } C_b = 1.81 \]

\[ S_{xc} = \text{Section modulus with respect to the compression flange} = \frac{448}{0.5 \cdot 15.86} = 56.5 \text{ in}^3 \]

\[ I_{yc} = \frac{tb^3}{12} = 0.43 \cdot 6.99^3/12 = 12.238 \text{ in}^4 \]

\[ J = (2 \cdot 6.99 \cdot 0.43^3 \cdot (15.86 - 2 \cdot 0.43) \cdot 0.29^3)/3 = 0.5 \text{ in}^4 \]

\[
F_{ez} = \frac{50(10)^6 1.81}{56.5} \left( \frac{12.238}{60} \right) \left[ 0.772 \frac{0.5}{12.238} + 9.87 \left( \frac{15}{60} \right)^2 \right] 10^{-3} = 263.1 \text{ ksi}
\]

which is larger than \( 0.55 \cdot F_Y = 19.8 \) ksi, so \( F_{cz} = 19.8 \) ksi

### Axial Compression

Critical \((kL/r) = 1.0 \cdot 60/1.52 = 69.69\)

As \((kL/r) < C_c\), the allow axial stress in compression is given by:

\[
F_a = \frac{F_Y}{F.S.} \left[ 1 - \frac{(kL/r)^2 F_Y}{4\pi^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(39.474)^2 36}{4\pi^2 29,000} \right] = 16.15 \text{ ksi}
\]

### Actual Stress

Actual axial stress, \( f_a = 14.02 / 10.6 = 1.323 \) ksi

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = \( f_{bz} = 57.04 \cdot 12/56.5 = 1.001 \cdot 12 = 12.115 \) ksi

\((KL/r)_x = 1 \cdot 60/6.5= 9.231\)

\[
F_{ez} = \frac{F_y}{F.S. (kL/r)^2} = \frac{\pi^2 E}{2.12(9.231)^2} = 1, 584.4 \text{ ksi}
\]

From Table 10-36A, \( C_{mz} = 0.85 \)

Equation 10-42
For the end section, use Equation 10.43:

\[
\frac{f_a}{F_a} + \frac{C_{mz}f_{bz}}{1 - \frac{f_a}{F'_{ez}}}F_{bz} + \frac{C_{my}f_{by}}{1 - \frac{f_a}{F'_{ey}}}F_{by} = \frac{1.323}{16.149} + \frac{0.85 \cdot 12.115}{1 - \frac{1.323}{1.584}} + 0 = 0.602
\]

The critical stress ratio is thus 0.689. The value calculated by STAAD is 0.690.

**Member 6**

Size W 16X36, L = 16 ft, a = 10.6 in², Sz = 56.5 in³

From observation Load case 3 will govern, Forces at the end are

- Fx = 10.2 kip (compression)
- Mz = 62.96 k-ft

Calculate the allowable stress as per Table 10.32.1A.

**Bending Minor Axis**

Allowable minor axis bending stress:

\[ F_{TY} = F_{TZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi} \]

**Bending Major Axis**

\[
F_{cz} = \frac{50(10)^6C_b}{S_{xc}} \left(\frac{I}{I_{yc}}\right) \sqrt{0.772 \frac{J}{I_{yc}}} + 9.87 \frac{d^4}{I} \leq 0.55F_Y
\]

Where:

\[
C_b = 1.75 + 1.05(M1/M2) + 0.3 \cdot (M1/M2)^2
\]

M1 = 8.947 M2 = 183.05 so C_b = 1.69

\[
S_{xc} = \text{Section modulus with respect to the compression flange} = \frac{448}{(0.5 \cdot 15.86)} = 56.5 \text{ in}^3
\]

\[
I_{YC} = \frac{tb^3}{12} = 0.43 \cdot 6.99^3/12 = 12.238 \text{ in}^4
\]

\[
J = (2 \cdot 6.99 \cdot 0.43^3 + (15.86 - 2 \cdot 0.43) \cdot 0.29^3)/3 = 0.5 \text{ in}^4
\]

\[
F_{cz} = \frac{50(10)^61.81}{56.5} \left(\frac{12.238}{192}\right) \sqrt{0.772 \cdot 0.5 \frac{12.238}{12.238} + 9.87 \left(\frac{15}{192}\right)^2} \leq 287.9 \text{ ksi}
\]

which is larger than 0.55 \cdot F_Y = 19.8 ksi, so F_{CZ} = 19.8 ksi

**Axial Compression**

Critical \((kL/r) = 1.0 \cdot 192/1.52 = 126.3\)

As \((kL/r) > C_c\), the allow axial stress in compression is given by:

\[
F_a = \frac{n^2E}{F.S. (kL/r)^2} = \frac{n^229, 000}{2.12(126.3)^2} = 8.46 \text{ ksi}
\]

**Actual Stress**
Actual axial stress, \( f_a = \frac{10.2}{10.6} = 0.962 \text{ ksi} \)

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = \( f_{bz} = \frac{62.96 \cdot 12}{56.5 \cdot 12} = 1.114 \cdot 12 = 13.37 \text{ ksi} \)

\[
\left(\frac{KL}{r}\right)_a = 1 \cdot \frac{192/6.51}{12} = 29.49
\]

\[
F_{ez} = \frac{n^2E}{F.S.(kL/r)^2} = \frac{n^229.000}{2.12(29.49)^2} = 155.2 \text{ ksi}
\]

From Table 10-36A, \( C_{mx} = 0.85 \)

Equation 10-42

\[
\frac{f_a}{F_a} + \frac{C_{mx}f_{bz}}{F_{ez}} + \frac{C_{my}f_{by}}{F_{by}} = \frac{0.962}{8.46} + \frac{0.85 \cdot 13.37}{19.8} + 0 = 0.691
\]

For the end section, use Equation 10.43:

\[
\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.962}{0.472(36)} + \frac{13.37}{19.8} + 0 = 0.732
\]

The critical stress ratio is thus 0.732. The value calculated by STAAD is 0.732

**Note:** The program gives this value when the slenderness check is suppressed (MAIN 1.0 for member 6); otherwise the member fails as a compression member with a slenderness parameter greater than 120).

**Member 7**

Size W 16X36, \( L = 4 \text{ ft}., a = 10.6 \text{ in}^2, Sz = 56.5 \text{ in}^3 \)

From observation Load case 3 will govern, Forces at the midspan are

- \( Fx = 24.05 \text{ kip (tension)} \)
- \( Mz= 62.96 \text{ k-ft} \)

Calculate the allowable stress as per Table 10.32.1A

**Bending Minor Axis**

Allowable minor axis bending stress:

\[ FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi} \]

**Bending Major Axis**

\[ FCZ = 0.55 \cdot F_Y = 19.8 \text{ ksi} \]

**Axial Tension**

**Note:** No connection information is specified and no reduction of section is assumed.

\[ F_a = 0.55 \cdot F_Y = 19.8 \text{ ksi} \]

**Actual Stress**

Actual axial stress, \( f_a = \frac{24.05}{10.6} = 2.268 \text{ ksi} \), hence, ok.
Actual bending stress = \( f_{bz} = 62.96 \cdot \frac{12}{56.5} = 1.114 \cdot 12 = 13.37 \text{ ksi} \)

So the combined ratio is

\[
\frac{f_a}{F_a} + \frac{f_{bz}}{F_{TZ}} + \frac{f_{by}}{F_{TY}} = \frac{2.268}{19.8} + 13.37/19.8 + 0 = 0.790
\]

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.43 in section 10-36 to calculate the design ratio for the end section.

\[
\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{2.268}{0.472(36)} + \frac{13.37}{19.8} + 0 = 0.809
\]

The critical stress ratio is thus 0.809. The value calculated by STAAD is 0.809.

**Member 8**

Size L4x4x1/4, \( L = 7.07 \text{ ft} \), \( a = 1.938 \text{ in}^2 \)

From observation Load case 1 will govern, Forces

\( F_x = 23.04 \text{ kip (compression)} \)

Calculate the allowable stress as per Table 10.32.1A

**Axial Compression**

Critical \((KL/r)_y = 1.0 \cdot 7.07 \cdot 12/0.795 = 106.7 \)

As \((kL/r) < C_c \), the allow axial stress in compression is given by:

\[
F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(kL/r)^2 F_y}{4n^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(106.7)^2}{36} \right] = 10.89 \text{ ksi}
\]

**Actual Stress**

Actual axial stress, \( f_a = 23.04 / 1.938 = 11.88 \text{ ksi} \)

\( f_a / F_a = 11.88/10.89 = 1.091 \)

The value calculated by STAAD is 1.091.

**Member 9**

Size L5x5x3/8, \( L = 5.657 \text{ ft} \), \( A = 3.61 \text{ in}^2 \)

From observation Load case 1 will govern, Forces

\( F_x = 48.44 \text{ kip (compression)} \)

Calculate the allowable stress as per Table 10.32.1A

**Axial Compression**

Critical \((KL/r)_y = 1.0 \cdot 5.567 \cdot 12/0.99 = 68.57 \)

As \((kL/r) < C_c \), the allow axial stress in compression is given by:

\[
F_a = \frac{F_y}{F.S.} \left[ 1 - \frac{(kL/r)^2 F_y}{4n^2 E} \right] = \frac{36}{2.12} \left[ 1 - \frac{(68.57)^2}{36} \right] = 14.47 \text{ ksi}
\]

**Actual Stress**

Verification Examples

STAAD.Pro 3837 User Manual
Actual axial stress, \( f_a = \frac{48.44}{3.61} = 13.42 \text{ ksi} \)

\[ f_a/F_a = \frac{13.42}{14.47} = 0.927 \]

The value calculated by STAAD is 0.928.

Comparison

**Table 535: Comparison of results**

<table>
<thead>
<tr>
<th>Member Number</th>
<th>STAAD.Pro Results</th>
<th>Hand Calculation</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.216</td>
<td>1.217</td>
<td>none</td>
</tr>
<tr>
<td>2</td>
<td>1.091</td>
<td>1.092</td>
<td>none</td>
</tr>
<tr>
<td>3</td>
<td>1.207</td>
<td>1.203</td>
<td>none</td>
</tr>
<tr>
<td>4</td>
<td>1.130</td>
<td>1.126</td>
<td>none</td>
</tr>
<tr>
<td>5</td>
<td>0.691</td>
<td>0.689</td>
<td>none</td>
</tr>
<tr>
<td>6</td>
<td>1.052</td>
<td>0.732</td>
<td>43.7% (^1)</td>
</tr>
<tr>
<td>7</td>
<td>0.812</td>
<td>0.808</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>8</td>
<td>1.114</td>
<td>1.091</td>
<td>2.1%</td>
</tr>
<tr>
<td>9</td>
<td>0.920</td>
<td>0.927</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

**Note:** \(^1\) The ratio is 0.832 when the slenderness check is suppressed, which results in a 13.7% difference.

**STAAD Input**

```
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AASHTO\AASHTO 17th Ed ASD - Design Frame.STD is typically installed with the program.

STAAD PLANE VERIFICATION PROBLEM FOR AASHTO CODE
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
*
* THIS DESIGN EXAMPLE IS VERIFIED BY HAND CALCULATION
* FOLLOWING AASHTO ASD 97 CODE.
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0; 3 0 10 0; 4 25 11 0; 5 0 15 0; 6 25 15 0; 7 5 15 0;
8 21 15 0;
MEMBER INCIDENCES
1 1 3; 2 3 5; 3 2 4; 4 4 6; 5 5 7; 6 7 8; 7 8 6; 8 3 7; 9 4 8;
MEMBER PROPERTY AMERICAN
1 2 TABLE ST W12X26
3 4 TABLE ST W14X43
```
5 TO 7 TABLE ST W16X36
8 TABLE ST L40404
9 TABLE ST L50506
MEMBER TRUSS
8 9
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 PINNED
LOAD 1 DL + LL
MEMBER LOAD
5 TO 7 UNI Y -2
LOAD 2 WIND FROM LEFT
JOINT LOAD
5 FX 15
LOAD COMBINATION 3
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT MEMBER FORCES
PARAMETER 1
CODE AASHTO
TRACK 0 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.Pro CODE CHECKING - (AASHTO - ASD) v1.0
*****************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/ LOCATION
FX MY MZ LOCATION
=======================================================================
* 1  ST  W12X26 (AISC SECTIONS)
FAIL  AASHTO 10-43 1.216 1
25.00 C 0.00 56.50 10.00
* 2  ST  W12X26 (AISC SECTIONS)
FAIL  AASHTO 10-43 1.091 1
8.72 C 0.00 56.50 0.00
* 3  ST  W14X43 (AISC SECTIONS)
FAIL  AASHTO 10-43 1.207 3
25.50 C 0.00 -112.20 11.00
* 4  ST  W14X43 (AISC SECTIONS)
FAIL  AASHTO 10-43 1.130 3
8.83 T 0.00 -112.20 0.00
5  ST  W16X36 (AISC SECTIONS)
PASS  AASHTO 10-43 0.691 3
14.02 C 0.00 -57.00 5.00
* 6  ST  W16X36 (AISC SECTIONS)
FAIL  KL/R ratio 1.052 1
5.65 C 0.00 -15.25 0.00
V. AASHTO 2nd Ed LRFD - Design Beam

The following compares the solution of a design performed using STAAD.Pro against a hand calculation of steel design per the AASHTO (LRFD) code.

Reference

The following step by step hand calculation as per AASHTO LRFD(1998) code.

Related Links

- [D1.D.2 AASHTO (LRFD)](on page 1468)

Problem

Determine the allowable resistances (AASHTO LRFD, 1998 code) for the member of the structure as shown in figure. Also, perform a code check for the member based on the results of the analysis.

A 48 foot long, non-composite girder is assumed to be simply supported. The section is a plate girder with 16" x 1" plate flanges and a 34" x 3.6" plate web (36 in. total depth). All plates are grade 36 steel.

The beam is subject to a 9.111 kip/ft uniform load.
Solution

\[ A = 154.4 \text{ in}^2, \; I_z = 21,594 \text{ in}^4, \; I_y = 814.9 \text{ in}^4 \]
\[ S_z = \frac{21,594(2)}{36} = 1,200 \text{ in}^3 \]
\[ r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{814.9}{154.4}} = 2.30 \text{ in} \]
\[ r_z = \sqrt{\frac{I_z}{A}} = \sqrt{\frac{21,594}{154.4}} = 11.83 \text{ in} \]

From observation Load case 1 will govern,
\[ M_z = 2,624 \text{ kip-ft} \]

Axial Compression Capacity

Refer Clause 6.9.4 of the code.

\[ (kL/r)_y = 0.333 \times \frac{576}{2.297} = 83.50 \]
\[ (kL/r)_z = \frac{1 \times 576}{11.826} = 48.71 \]
\[ (kL/r)_{\text{crit}} = 83.50 < 120, \text{ ok.} \]

Calculation of Width/Thickness ratio for axial compression

Plate buckling coefficients taken from Table 6.9.4.2-1:
\[ k_w = 1.49, \; k_f = 0.56 \]

Slenderness ratio for the web:
\[ \left( d - 2 \cdot t_f \right)/t_w = \frac{36 - 2 \cdot 1}{3.6} = 9.444 \]

Slenderness ratio for the 1/2 flange:
\[ b_f/(2 \cdot t_f) = \frac{16}{2 \cdot 1} = 8.0 \]

Critical ratio for web:
Critical ratio for flange:

\[ k_f \sqrt{\frac{E}{F_y}} = 0.56 \sqrt{\frac{29,000}{36}} = 15.89 > 8.0 \]

Thus, OK [AASHTO LRFD Cl. 6.4.9.2]

Slenderness ratio about major and minor axis:

\[ \lambda_z = \left( \frac{Kl}{r_z} \right) \frac{F_y}{E} = \left( \frac{1.0 \cdot 576}{11.83} \right)^{\frac{2}{3}} \frac{36}{29,000} = 0.298 \]

\[ \lambda_y = \left( \frac{Kl}{r_y} \right) \frac{F_y}{E} = \left( \frac{0.333 \cdot 576}{2.30} \right)^{\frac{2}{3}} \frac{36}{29,000} = 0.877 \]

\( \lambda_y \) governs, thus \( \lambda = 0.877 \)

\( \lambda < 2.25 \), so Equation 6.9.4.1-1 is used to determine the nominal compressive resistance

\[ P_n = 0.66 \lambda F_y A_s = 0.66 \cdot 0.877 \cdot 36 \cdot 154.4 = 3,861 \text{ kips} \]

The factored compressive resistance,

\[ P_r = \phi c P_n = 0.9 \cdot 3,861 = 3,475 \text{ kips} \]

**Major Axis Bending Capacity**

The compression flange moment of inertia:

\[ I_{yc} = 1(16)^\frac{3}{12} = 341.3 \text{ in}^4 \]

\[ I_{yc}/I_y = 341.3/814.9 = 0.419, > 0.1 \text{ and } < 0.9 \]

Thus, OK [AASHTO LRFD Cl. 6.10.2.1]

\[ \frac{2D_6}{t_w} = \frac{2(17)}{3.6} = 9.445 < 6.77 \sqrt{\frac{E}{F_y}} = 6.77 \sqrt{\frac{29,000}{36}} = 192.1 \]

Thus, OK [AASHTO LRFD Cl. 6.10.2.2]

Calculation of depth of the web in compression at the plastic moment, \( D_{cp} \) (Clause 6.10.3.3.2)

Area of Web, \( A_w = (36 - 2 \times 1) \times 3.6 = 122.4 \text{ in}^2 \)

Area of flange area in tension, \( A_{ft} = 16 \times 1 = 16 \text{ in}^2 \)

\[ D_{cp} = \left( \frac{D_w}{2A_w} \right) F_y \left( A_{ft} + A_w - A_{fc} \right) = \left( \frac{34}{2 \cdot 122.4 \cdot 36} \right) 36 \left( 16 + 122.4 - 16 \right) = 17 \text{ in} \]

\[ \frac{2D_{cp}}{t_w} = \frac{2(17)}{3.6} = 9.445 < 3.76 \sqrt{\frac{E}{F_y}} = 3.76 \sqrt{\frac{29,000}{36}} = 108.1 \]

Thus, OK [AASHTO LRFD Cl. 6.10.4.1.2]

\[ \frac{b_f}{t_f} = \frac{16.0}{21.0} = 0.80 < 0.382 \sqrt{\frac{E}{F_y}} = 0.382 \sqrt{\frac{29,000}{36}} = 10.98 \]

Thus, OK [AASHTO LRFD Cl. 6.10.4.3]

Check clause 6.10.4.1.6a
\[(B/t)_{\text{flange}} < 0.75 \times (B/t)_{\text{flange\_limit}} = 0.75 \times 10.978\]
\[(D/T)_{\text{web}} < 0.75 \times (D/T)_{\text{web\_limit}} = 0.75 \times 108.057\]

Check clause 6.10.4.1.7

\[M_{pz} = P_z \times F_y = 1,600 \text{ in}^3(36 \text{ ksi}) = 57,614 \text{ in-k}\]

\[L_b = \left[0.124 - 0.0759\left(\frac{M_l}{M_{pz}}\right)\right] \times \left[\frac{r_y E}{F_y}\right] = 0.124 - 0.0759\left(\frac{0}{57,614}\right) \times \left[\frac{2.30(29,000)}{36}\right] = 229.5 \text{ in}\]

Unsupported length, \(L_u = 576\) in > \(L_b\); section is non-compact.

Check clause 6.10.4.1.9

A notional section comprised of the compression flange and one-third of the depth of the web in compression, taken about the vertical axis.

\[A_{rt} = A_{cf} + \left(\frac{c - t_f}{3}\right)t_w = 16 + \left(\frac{18 - 1.0}{3}\right)3.6 = 36.4 \text{ in}^2\]

\[I_{rt} = t_f \left[\left(\frac{c - t_f}{3}\right)t_w\right] = 1\left[\left(\frac{18 - 1}{3}\right)\frac{3.6^3}{12}\right] = 363.4 \text{ in}^4\]

\[r_{rt} = \sqrt{\frac{363.4}{36.4}} = 3.149 \text{ in}\]

\[L_p = 1.76\left(3.159\right)\sqrt{\frac{29,000}{36}} = 159.8 \text{ in}\]

\[L_b > L_p\]

Clause 6.10.4.2.6

\[L_r = 4.44\sqrt{\frac{I_{yc}}{S_{xc} \frac{E}{F_y}}} = 4.44\sqrt{\frac{341.3(36)}{1,200 \frac{29,000}{36}}} = 403.3 \text{ in}\]

Minimum radius of gyration of the compression flange taken about the vertical axis:

\[r_l = \sqrt{\frac{I_{yc}}{A_{cf}}} = \sqrt{\frac{341.3}{16}} = 4.619 \text{ in}\]

Hybrid factor, \(R_h = 1.0\) (For Homogeneous sections, Hybrid factors shall be taken as 1.0 per clause 6.10.4.3.1)

As per clause 6.10.4.3.2, Load-shedding factor, \(R_b\)

If area of the compression flange, \(A_{cf} \geq\) the area of the tension flange, \(A_{tf}\)

\[l_f = 5.76\]

\[D_c = 17.00\]

\[\frac{2D_c}{t_w} = \frac{2(17)}{3.6} = 9.445 < L_b\sqrt{\frac{E}{F_y}} = 5.76\sqrt{\frac{29,000}{36}} = 163.5\]

Thus:

\[R_{b,\text{comp}} = R_{b,\text{ten}} = 1.0\]

\[L_r < L_b\]

\[C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2\]

Here \(M1 = 0, M2 = 0\) so \(C_b = 1.75\)
Verification Examples
V.09 Steel Design

\[ M_{nz,\text{comp}} = C_{bz} R_{b,\text{comp}} R_h \frac{M_y \left( \frac{L_r}{L_b} \right)^2}{L_b} = 1.75 \left( \frac{1.0}{1.0} \right) \left( \frac{36(1.200)}{576} \right)^2 = 19,000 \text{kip} \cdot \text{in} \]

\[ M_{nz,\text{ten}} = R_{b,\text{ten}} R_h F_y Z = 1.0(1.0)(36)(1, 200) = 43,188 \text{kip} \cdot \text{in} \]

\[ M_{nz} = 19,000 \text{kip in} \]

Resisting Moment

\[ M_r = Q_f \cdot M_n = 1.0 (19,000) = 19,000 \text{kip in} \]

Actual Moment = 31,488 kip\cdot in

Interaction ratio = 31,488 / 19,000 = 1.657

Comparison

Table 536: Comparison of results

<table>
<thead>
<tr>
<th>Value</th>
<th>Hand Calculations</th>
<th>STAAD.Pro Results</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Interaction Ratio</td>
<td>1.657</td>
<td>1.654</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The following input is used in this verification example.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-May-11
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 38 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
AASHTOGIRDER
154.4 36 3.6 16 1 21593.8 814.86 539.4 129.6 32
END
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 AASHTOGIRDER
Verification Examples
V.09 Steel Design

CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
UNIT INCHES KIP
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.76
PERFORM ANALYSIS
PARAMETER 1
CODE AASHTO LRFD
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

PAGE NO.
1
******************************************************************************************************************************************
*                                                                                                                                   *
*          STAAD.Pro CONNECT Edition                                                *
*          Version 22.01.00.**                                                    *
*          Proprietary Program of                                                *
*          Bentley Systems, Inc.                                                 *
*          Date= APR 14, 2019                                                    *
*          Time= 23:10:48                                                        *
*                                                                                                                                   *
* Licensed to: Bentley Systems Inc                                              *
******************************************************************************************************************************************

1. STAAD SPACE
INPUT FILE: AASHTO 2nd Ed LRFD - Design Beam.STD
2. START JOB INFORMATION
3. ENGINEER DATE 23-MAY-11
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT FEET KIP
7. JOINT COORDINATES
   1 0 0 0; 2 38 0 0
8. MEMBER INCIDENCES
10. 1 1 2
11. START USER TABLE
12. TABLE 1
13. UNIT INCHES KIP
14. WIDE FLANGE
15. AASHTOGIRDER
16. 154.4 36 3.6 16 1 21593.8 814.86 539.4 129.6 32
17. END
18. UNIT FEET KIP
19. DEFINE MATERIAL START
20. ISOTROPIC STEEL
21. E 29000
22. POISSON 0.3
23. DENSITY 0.000283
24. ALPHA 6E-06
25. DAMP 0.03
26. TYPE STEEL
27. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
28. END DEFINE MATERIAL
29. MEMBER PROPERTY AMERICAN
30. 1 UPTABLE 1 AASHTOGIRDER
31. CONSTANTS
32. MATERIAL STEEL ALL
33. SUPPORTS
34. 1 PINNED
35. 2 FIXED BUT FX MY MZ
36. UNIT INCHES KIP
37. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
38. MEMBER LOAD
   STAAD SPACE
   -- PAGE NO.
2
39. 1 UNI GY -0.76
40. PERFORM ANALYSIS
** WARNING ** A SOFT MATERIAL WITH (1.0 / 1.564E+01) TIMES THE STIFFNESS OF CONCRETE ENTERED. PLEASE CHECK.

 PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       6
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
41. PARAMETER 1
42. CODE AASHTO LRFD
43. TRACK 2 ALL
44. CHECK CODE ALL
STEEL DESIGN
   STAAD SPACE
   -- PAGE NO.
3

STAAD.PRO CODE CHECKING - ( AASHTO - LRFD) v1.0
********************************************************************************
*  1 ST AASHTOGIRDER             (UPT)
FAIL  Slenderness  1.654   1
     0.00   0.00   0.00
Section Properties (in)
-----------------------
Ax =     154.40   Ay =     129.60   Az =      32.00
Iz =   21593.80   Iy =     814.86
Rz =      11.83   Ry =       2.30
Sz =    1199.66   Sy =     101.86
Input Parameters (Kip-in)
--------------------------
Fyld  =      36.00   Fu    =      58.00
Lz    =     456.00   Ly    =     456.00
Kz    =       1.00   Ky    =       1.00
UNL   =       0.00   Ratio =       1.00
Design Results (Kip-in)
-----------------------
Klrz =      38.56   Klry =     198.49
CBy  =       0.00   CBz  =       0.00
CAPACITIES (Kip-in)
---------------------
V. AISC

V. AISC 360-2016

V. Bending

V. AISC 360-16 W Flexural Check LRFD

Verify the flexural yielding strength and lateral torsional buckling of an I section using the LRFD method per AISC 360-16.

References


Details

Verify the flexural yielding strength and lateral torsional buckling of an I section (W12X90) gr. ASTM A992 material, with a member length of 41'-8", and a concentrated moment of 35,000 in-kip applied at midspan. The member is unbraced along its length.

Validation

Flexural yielding about X axis

As the section is compact, nominal flexural yielding strength about X axis is:

\[ M_{nx} = f_y \times Z_{xx} = 65 \text{ ksi} \times 311 \text{ in}^3 = 20,215 \text{ in-kip} \]  \hspace{1cm} (F2-1, ref. 1)
\[ M_x = \phi_c \times M_{nx} = 0.9(20,215 \text{ in·kip}) = 18,193.5 \text{ in·kip} \]

**Flexural yielding about Y axis**

As this is the minor axis, nominal flexural yielding strength about Y axis is:

\[ M_{ny} = f_y \times Z_{yy} = 65 \text{ ksi} \times 143 \text{ in}^3 = 9,295 \text{ in·kip} \]

\[ M_y = \phi_c \times M_{ny} = 0.9(9,295 \text{ in·kip}) = 8,365.5 \text{ in·kip} \]  

**Lateral torsional buckling about X axis**

\[ r_y = \sqrt{I_{yy}/A_g} = 3.243 \text{ in}^2 \]

Limiting laterally unbraced length for the yielding limit state:

\[ L_p = 1.76 \times r_y \frac{E}{F_y} = 120.56\text{in} \]  

\[ r_{ts} = \sqrt{I_{yy} \times C_w}{S_{xx}} = 3.769\text{in} \]

For an I section, \( C = 1 \) (per F2-28a in ref. 1). The limiting unbraced length for the inelastic lateral-torsional buckling limit state is:

\[ h_0 = d - t_f = 12.66 \text{in} \]

\[ L_r = 1.95 \times r_{ts} \frac{E}{0.7F_y} \sqrt{\frac{J_c}{S_{xx}h_0} + \sqrt{\left(\frac{J_c}{S_{xx}h_0}\right)^2 + 6.76\left(\frac{0.7F_y}{E}\right)^2}} = 810.27\text{in} \]

\[ L_p < L_b = 500 \text{ in} < L_r \]

\[ C_b = \frac{12.5M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_c} = \frac{12.5(20,000)}{2.5(20,000) + 3(10,000) + 4(20,000) + 3(10,000)} = 1.316 \]

\[ M_p = f_y \times Z_{xx} = 65 \text{ ksi} \times 311 \text{ in}^3 = 20,215 \text{ in·kip} \]

Nominal LTB strength:

\[ M_n = C_b \left[ M_p - \left( M_p - 0.7F_y S_{Sy} \right) \frac{L_b - L_r}{L_p} \right] \leq M_p \]

\[ M_n = 20,611.4 \text{ in·kip} > M_p \]

\( \phi_c M_{nx} = 0.9(20,215 \text{ in·kip}) = 18,193.5 \text{ in·kip} \)

**Results**

**Table 537: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexural yielding capacity, X axis (in·kip)</td>
<td>18,194</td>
<td>18,190</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Flexural buckling capacity, Y axis (in·kip)</td>
<td>8366</td>
<td>8366</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Parameter</td>
<td>Hand Calculation</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>----------------------------------------</td>
<td>------------------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>Lateral-torsional buckling about X axis (in-kip)</td>
<td>18,194</td>
<td>18,190</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2016\Bending\AISC 360-16 W Flexural Check LRFD.STD is typically installed with the program.

STAAD SPACE

START JOB INFORMATION
ENGINEER DATE 05-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 41.67 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
type steel
strength fy 5184 fu 8352 ry 1.5 rt 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 table st w12x190
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT MY
UNIT INCHES KIP
LOAD 2 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 Cmom gz 35000 250
UNIT FEET KIP
PERFORM ANALYSIS
PARAMETER 1
CODE AISC UNIFIED 2016
LY 41.6667 ALL
LZ 41.6667 ALL
FU 9360 ALL
FYLD 9360 ALL
METHOD LRFD
PROFILE W12 ALL
STP 1 ALL
TRACK 2 ALL
UNIT KIP INCH
### Verification Examples

**V.09 Steel Design**

------------------------------

**CHECK CODE ALL**

**FINISH**

**STAAD Output**

---

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

******************************************************************************

**ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).**

*****NOTE : AISC 360-16 Design Statement for STAAD.Pro.***

***** AXIS CONVENTION ***:**

The capacity results and intermediate results in the report follow the notations and axes labels as defined in the AISC 360-16 code. The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16 design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

**SECTION FORCES AXIS MAPPING:**

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

***** DESIGN MESSAGES ***:**

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis.
   
   E.g : Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

***** ABBREVIATIONS ***:**

------------------

F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
F-L-B = Flange Local Buckling
W-L-B = Web Local Buckling
**Member:**  1  
**Profile:**  ST W12X190  
**Status:**  PASS  
**Location:**  208.35  

**SLENDERNESS**

- **Actual Slenderness Ratio:** 154.185
- **Allowable Slenderness Ratio:** 300.000  

**STRENGTH CHECKS**

- **Critical L/C:** 2  
- **Ratio:** 0.903  
- **Loadcase:** 2

**SECTION PROPERTIES**  
(LOC: 208.35, PROPERTIES UNIT: IN )

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>5.600E+01</td>
</tr>
<tr>
<td>Axx</td>
<td>4.420E+01</td>
</tr>
<tr>
<td>Ayy</td>
<td>1.526E+01</td>
</tr>
<tr>
<td>Ixx</td>
<td>1.890E+03</td>
</tr>
<tr>
<td>Iyy</td>
<td>5.890E+02</td>
</tr>
<tr>
<td>J</td>
<td>4.880E+01</td>
</tr>
<tr>
<td>Sxx+</td>
<td>2.625E+02</td>
</tr>
<tr>
<td>Sxx-</td>
<td>2.625E+02</td>
</tr>
<tr>
<td>Zxx</td>
<td>3.110E+02</td>
</tr>
<tr>
<td>Syy+</td>
<td>9.276E+01</td>
</tr>
<tr>
<td>Syy-</td>
<td>9.276E+01</td>
</tr>
<tr>
<td>Zyy</td>
<td>1.430E+02</td>
</tr>
<tr>
<td>Cw</td>
<td>2.380E+04</td>
</tr>
<tr>
<td>x0</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>y0</td>
<td>8.882E-16</td>
</tr>
</tbody>
</table>

**MATERIAL PROPERTIES**

- **Fyld:** 65.000  
- **Fu:** 65.000
### Actual Member Length: 500.040

<table>
<thead>
<tr>
<th>Design Parameters</th>
<th>(Rolled)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kx: 1.00  Ky: 1.00  NSF: 1.00  SLF: 1.00  CSP: 12.00</td>
<td></td>
</tr>
</tbody>
</table>

#### COMPRESSION CLASSIFICATION (L/C: 2 LOC: 500.04)

<table>
<thead>
<tr>
<th>CASE</th>
<th>l</th>
<th>l p</th>
<th>l r</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange: NonSlender</td>
<td>3.65</td>
<td>N/A</td>
<td>11.83</td>
</tr>
<tr>
<td>Web : NonSlender</td>
<td>10.30</td>
<td>N/A</td>
<td>31.47</td>
</tr>
</tbody>
</table>

#### FLEXURE CLASSIFICATION (L/C: 1 LOC: 500.04)

<table>
<thead>
<tr>
<th>CASE</th>
<th>l</th>
<th>l p</th>
<th>l r</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange: Compact</td>
<td>3.65</td>
<td>8.03</td>
<td>21.12</td>
</tr>
<tr>
<td>Web : Compact</td>
<td>10.30</td>
<td>79.42</td>
<td>120.40</td>
</tr>
</tbody>
</table>

#### TENSILE YIELDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>3276.00</td>
<td>0.000</td>
<td>C1.D2</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom. Ten. Yld Cap : Pn = 3640.0 kip  Eq.D2-1

#### TENSILE RUPTURE

| DEMAND | CAPACITY | RATIO | REFERENCE | L/C | LOC |

STAAD SPACE -- PAGE NO. 5

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

- Member : 1 Contd.

CHECKS FOR AXIAL TENSION

### STAAD Space
Intermediate Results:

Effective area : \( Ae = 56.000 \text{ in}^2 \)  \( \text{Eq.D3-1} \)

Nom. Ten. Rpt Cap : \( P_n = 3640.0 \text{ kip} \)  \( \text{Eq.D2-2} \)

CHECKS FOR AXIAL COMPRESSION

FLEXURAL BUCKLING X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>1620.0</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Effective Slenderness : \( L_{cx}/r_x = 86.066 \)  \( \text{Cl.E2} \)

Elastic Buckling Stress : \( F_{ex} = 38.639 \text{ ksi} \)  \( \text{Eq.E3-4} \)

Crit. Buckling Stress : \( F_{crx} = 32.146 \text{ ksi} \)  \( \text{Eq.E3-2} \)

Nom. Flexural Buckling : \( P_{nx} = 1800.2 \text{ kip} \)  \( \text{Eq.E3-1} \)

FLEXURAL BUCKLING Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>532.2</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Effective Slenderness : \( L_{cy}/r_y = 154.17 \)  \( \text{Cl.E2} \)

Elastic Buckling Stress : \( F_{ey} = 12.042 \text{ ksi} \)  \( \text{Eq.E3-4} \)

Crit. Buckling Stress : \( F_{cry} = 10.560 \text{ ksi} \)  \( \text{Eq.E3-3} \)

Nom. Flexural Buckling : \( P_{ny} = 591.39 \text{ kip} \)  \( \text{Eq.E3-1} \)

FLEX-TOR-BUCKLING
<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>2911.</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Elastic F-T-B Stress : \( \text{Fe} = 230.56 \text{ ksi} \) \( \text{Eq.E4-2} \)

Crit. F-T-B Stress : \( \text{Fcr} = 57.765 \text{ ksi} \) \( \text{Eq.E3-2} \)

Nom. Flex-tor Buckling : \( \text{Pn} = 3234.9 \text{ kip} \) \( \text{Eq.E4-1} \)

---

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>1551.</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Coefficient Cv Along X : \( \text{Cv} = 1.0000 \) \( \text{Eq.G2-9} \)

Coefficient Kv Along X : \( \text{Kv} = 1.2000 \) \( \text{C1.G6} \)

Nom. Shear Along X : \( \text{Vnx} = 1723.6 \text{ kip} \) \( \text{Eq.G6-1} \)

---

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>78.85</td>
<td>595.3</td>
<td>0.132</td>
<td>Cl.G1</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Coefficient Cv Along Y : \( \text{Cv} = 1.0000 \) -
<table>
<thead>
<tr>
<th>Coefficient Kv Along Y</th>
<th>Kv     =  5.3400                Eq.G2-5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nom. Shear Along Y</td>
<td>Vny    =  595.30     kip          Eq.G2-1</td>
</tr>
</tbody>
</table>

**STAAD SPACE**

ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).

---

**CHECKS FOR BENDING**

**FLEX YIELDING (X)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>-.1643E+05</td>
<td>0.1819E+05</td>
<td>0.903</td>
<td>Cl.F2.1</td>
<td>2</td>
<td>208.35</td>
</tr>
</tbody>
</table>

Intermediate Results :

Nom Flex Yielding Along X : Mnx    =  20215.     kip-in     Eq.F2-1

**FLEX. YIELDING (Y)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>8366.</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results :

Nom Flex Yielding Along Y : Mny    =  9295.0     kip-in     Eq.F6-1

**LAT TOR BUCK ABOUT X**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>-.1643E+05</td>
<td>0.1819E+05</td>
<td>0.903</td>
<td>Cl.F2.2</td>
<td>2</td>
<td>208.35</td>
</tr>
</tbody>
</table>

Intermediate Results :

Nom L-T-B Cap             : Mnx    =  20215.     kip-in     Eq.F2-2
V. Compression

V. AISC 360-16 I Compression LRFD
Verify the flexural buckling capacity of an I section under compressive load using the LRFD method per the AISC 360-16 code.

References

Details
Verify the Flexural Buckling strength of an I Section (W14x90) using the LRFD method

Steel Gr. ASTM A992, about X and Y axis with a member length of 30 ft with dead load 140 kips and Live Load 420 kips.

Validation

Strong axis unbraced length, \( L_x = 360 \text{ in} \)

Torsional axis unbraced length \( L_y = 180 \text{ in} \)

Ultimate load, \( P_u = 1.2(180 \text{ kips}) + 1.6(420 \text{ kips}) = 840 \text{ kips} \)

**Calculation for major axis (X axis)**

From Table C-A-7.1 of ref. 1, \( K = 1.0 \) for a pinned-pinned condition.

\[
\frac{b_f}{2t_f} = 10.211 < 0.56\sqrt{\frac{E}{F_y}} = 13.487 \text{, thus the flange is non-slender.}
\]

Slenderness ratio about X axis \( \frac{K L_x}{r_x} = 58.632 < 4.71\sqrt{\frac{E}{F_y}} = 113.432 \)

Calculate the elastic critical buckling stress: \( F_{eX} = \frac{\pi^2 E}{(K L_x / r_x)^2} = 83.259 \text{ksi} \)

Calculate the flexural buckling stress: \( F_{cX} = 0.658 \frac{F_y}{F_{eX}} F_y = 38.887 \text{ksi} \)

Compressive strength for flexural buckling along X: \( \phi_c P_{nBX} = \phi_c F_{cX} A_g = 947.462 \text{kips} \)

Ratio for flexural buckling strength = 0.906

**Calculation for minor axis (Y axis)**

\[
h = d - 2 \cdot t_f = 12.58 \text{ in}
\]

\[
\frac{h}{w} = 28.591 < 35.884 \text{, thus the flange is non-slender.}
\]

Slenderness ratio about Y axis \( \frac{K L_y}{r_y} = 48.703 < 4.71\sqrt{\frac{E}{F_y}} = 113.432 \)

Calculate the elastic critical buckling stress: \( F_{eY} = \frac{\pi^2 E}{(K L_y / r_y)^2} = 120.668 \text{ksi} \)

Calculate the flexural buckling stress: \( F_{cY} = 0.658 \frac{F_y}{F_{eY}} F_y = 42.039 \text{ksi} \)

Compressive strength for flexural buckling along Y: \( \phi_c P_{nBY} = \phi_c F_{cY} A_g = 1, 002.625 \text{kips} \)

Ratio for flexural buckling strength = 0.838

Results
### Table 538: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexural Buckling Capacity, X Axis (kips)</td>
<td>927.5</td>
<td>927.5</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Ratio</td>
<td>0.906</td>
<td>0.906</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Flexural Buckling Capacity, Y Axis (kips)</td>
<td>1,003</td>
<td>1,003</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Ratio</td>
<td>0.838</td>
<td>0.838</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2016\Compression\AISC 360-16 I Compression LRFD.STD is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Jan-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 9360 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
JOINT LOAD
2 FY -140
LOAD 2 LOADTYPE Live REDUCIBLE TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 COMBINATION LOAD CASE 3
```
PERFORM ANALYSIS
LOAD LIST 3
*CODE AISC UNIFIED 2016
PARAMETER 1
CODE AISC UNIFIED 2016
FU 9360 ALL
FYLD 7200 ALL
KX 1 ALL
KY 1 ALL
KZ 1 ALL
LX 30 ALL
LY 15 ALL
LZ 30 ALL
METHOD LRFD
PROFILE W14 ALL
STP 1 ALL
*SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

--- STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1) ---
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:
The capacity results and intermediate results in the report follow the
notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING: -

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:
1. Section classification reported is for the cross section and loadcase that
produced the worst case design ratio for flexure/compression Capacity results.

2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.

3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis.
   E.g.: Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
F-L-B = Flange Local Buckling
W-L-B = Web Local Buckling
L-L-B = Leg Local Buckling
C-F-Y = Compression Flange Yielding
T-F-Y = Tension Flange Yielding

<table>
<thead>
<tr>
<th>Member No: 1</th>
<th>Profile: ST W14X90</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status: FAIL</td>
<td>Ratio: 1.004</td>
<td>Loadcase: 3</td>
</tr>
<tr>
<td>Location: 0.00</td>
<td>Ref: Cl.E4</td>
<td></td>
</tr>
<tr>
<td>Pz: 840.0 C</td>
<td>Vy: 0.000</td>
<td>Vx: 0.000</td>
</tr>
<tr>
<td>Tz: 0.000</td>
<td>My: 0.000</td>
<td>Mx: 0.000</td>
</tr>
</tbody>
</table>

SLENDERNESS

Actual Slenderness Ratio : 58.633
Allowable Slenderness Ratio : 200.000 LOC : 0.00

STRENGTH CHECKS

Critical L/C : 3 Ratio : 1.004(FAIL)
Loc : 0.00 Condition : Cl.E4
### SECTION PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>$2.650 \times 10^1$</td>
</tr>
<tr>
<td>Axx</td>
<td>$2.059 \times 10^1$</td>
</tr>
<tr>
<td>Ayy</td>
<td>$6.160 \times 10^0$</td>
</tr>
<tr>
<td>Ixx</td>
<td>$9.990 \times 10^2$</td>
</tr>
<tr>
<td>Iyy</td>
<td>$3.620 \times 10^2$</td>
</tr>
<tr>
<td>J</td>
<td>$4.060 \times 10^0$</td>
</tr>
<tr>
<td>Sxx+</td>
<td>$1.427 \times 10^2$</td>
</tr>
<tr>
<td>Sxx-</td>
<td>$1.427 \times 10^2$</td>
</tr>
<tr>
<td>Zxx</td>
<td>$1.570 \times 10^2$</td>
</tr>
<tr>
<td>Syy+</td>
<td>$4.993 \times 10^1$</td>
</tr>
<tr>
<td>Syy-</td>
<td>$4.993 \times 10^1$</td>
</tr>
<tr>
<td>Zyy</td>
<td>$7.560 \times 10^1$</td>
</tr>
<tr>
<td>Cw</td>
<td>$1.593 \times 10^4$</td>
</tr>
<tr>
<td>x0</td>
<td>$0.000 \times 10^0$</td>
</tr>
<tr>
<td>y0</td>
<td>$0.000 \times 10^0$</td>
</tr>
</tbody>
</table>

### MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld</td>
<td>$7200.000$</td>
</tr>
<tr>
<td>Fu</td>
<td>$9359.999$</td>
</tr>
</tbody>
</table>

### Actual Member Length

- **30.000**

### Design Parameters

- **Kx**: 1.00
- **Ky**: 1.00
- **NSF**: 1.00
- **SLF**: 1.00
- **CSP**: 1.00

### COMPRESSION CLASSIFICATION

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonSlender</th>
<th>10.21</th>
<th>N/A</th>
<th>13.49</th>
<th>Table.4.1a.Case1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Web</td>
<td>NonSlender</td>
<td>28.59</td>
<td>N/A</td>
<td>35.88</td>
<td>Table.4.1a.Case5</td>
</tr>
</tbody>
</table>

### FLEXURE CLASSIFICATION

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonCompact</th>
<th>10.21</th>
<th>9.15</th>
<th>24.08</th>
<th>Table.4.1b.Case10</th>
</tr>
</thead>
<tbody>
<tr>
<td>Web</td>
<td>Compact</td>
<td>28.59</td>
<td>90.55</td>
<td>137.27</td>
<td>Table.4.1b.Case15</td>
</tr>
</tbody>
</table>

---

**STAAD SPACE** -- PAGE NO. 5

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

- **Member**: 1 Contd.

**CHECKS FOR AXIAL TENSION**
**TENSILE YIELDING**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>1192.</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom. Ten. Yld Cap : Pn = 1325.0 kip Eq.D2-1

**TENSILE RUPTURE**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>1292.</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Effective area : Ae = 0.18403 ft<sup>2</sup> Eq.D3-1

**CHECKS FOR AXIAL COMPRESSION**

**FLEXURAL BUCKLING X**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>840.0</td>
<td>927.5</td>
<td>0.906</td>
<td>Cl.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Effective Slenderness : Lcx/rx = 58.633 Cl.E2
Elastic Buckling Stress : Fex = 11989. kip/ft<sup>2</sup> Eq.E3-4
Crit. Buckling Stress : Fcrx = 5599.7 kip/ft<sup>2</sup> Eq.E3-2
Nom. Flexural Buckling : Pnx = 1030.5 kip Eq.E3-1

**FLEXURAL BUCKLING Y**
## DEMAND/CAPACITY RATIO

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>840.0</td>
<td>1003.0</td>
<td>0.838</td>
<td>Cl.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results:**

- Effective Slenderness: \( \frac{L_{cy}}{r_{y}} = 48.701 \) (Cl.E2)
- Elastic Buckling Stress: \( \sigma_{ey} = 17377.0 \) kip/ft² (Eq.E3-4)
- Critical Buckling Stress: \( \sigma_{cry} = 6053.6 \) kip/ft² (Eq.E3-2)
- Nominal Flexural Buckling: \( P_{ny} = 1114.0 \) kip (Eq.E3-1)

## FLEX-TOR-BUCKLING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>840.0</td>
<td>837.0</td>
<td>1.004</td>
<td>Cl.E4</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results:**

- Elastic F-T-B Stress: \( \sigma_{e} = 8513.5 \) kip/ft² (Eq.E4-2)
- Critical F-T-B Stress: \( \sigma_{cr} = 5053.6 \) kip/ft² (Eq.E3-2)
- Nominal Flex-tor Buckling: \( P_{n} = 930.01 \) kip (Eq.E4-1)

---

**STAAD SPACE**

---

---

**Checks for Shear**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>555.9</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>
Coefficient Cv Along X : Cv     =  1.0000                Eq.G2-9
Coefficient Kv Along X : Kv     =  1.2000                Cl.G6
Nom. Shear Along X        : Vnx    =  617.70     kip        Eq.G6-1

SHEAR ALONG Y

Demands | Capacity | Ratio | Reference | L/C | LOC
---------|----------|-------|-----------|-----|-----
0.000    | 184.8    | 0.000 | Cl.G1     | 3   | 0.00

Intermediate Results :
Coefficient Cv Along Y : Cv     =  1.0000                -
Coefficient Kv Along Y : Kv     =  5.3400                Eq.G2-5
Nom. Shear Along Y        : Vny    =  184.80     kip        Eq.G2-1

FLEX. YIELDING (Y)

Demand | Capacity | Ratio | Reference | L/C | LOC
---------|----------|-------|-----------|-----|-----
0.000    | 283.5    | 0.000 | Cl.F6.1   | 3   | 0.00

Intermediate Results :
Nom. Flex Yielding Along Y : Mny    =  315.00     kip-ft     Eq.F6-1

LAT TOR BUCK ABOUT X

Demand | Capacity | Ratio | Reference | L/C | LOC
---------|----------|-------|-----------|-----|-----
0.000    | 465.8    | 0.000 | Cl.F2.2   | 3   | 0.00
Intermediate Results:
Nom L-T-B Cap : Mnx = 517.55 kip-ft Eq.F2-2
Mom. Distr. factor : CbX = 1.0000 Eq.F1-1
Limiting Unbraced Length : LpX = 13.055 ft Eq.F2-5
Coefficient C : Cx = 1.0000 Eq.F2-8a
Effective Rad. of Gyr. : Rts = 0.34183 ft Eq.F2-7
Limiting Unbraced Length : LrX = 42.564 ft Eq.F2-6

FLANGE LOCAL BUCK(X)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>573.6</td>
<td>0.000</td>
<td>Cl.F3.1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom F-L-B Cap : Mnx = 637.28 kip-ft Eq.F3-1

FLANGE LOCAL BUCK(Y)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>272.7</td>
<td>0.000</td>
<td>Cl.F6.2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom F-L-B Cap : Mny = 302.98 kip-ft Eq.F6-2

STAAD SPACE -- PAGE NO. 8

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
******************************************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted). - Member : 1 Contd.

CHECKS FOR AXIAL BEND INTERACTION
Intermediate Results:

<table>
<thead>
<tr>
<th>CLAUSE H1</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.004</td>
<td>Eq.H1-1a</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Axial Capacity: \( P_c = 837.01 \text{ kip} \)  C1.H1.1

Moment Capacity: \( M_{cx} = 465.79 \text{ kip-ft} \)  C1.H1.1

Moment Capacity: \( M_{cy} = 272.68 \text{ kip-ft} \)  C1.H1.1

---

**V. Shear**

V. AISC 360-16 Shear Strong Axis

Verify the shear strength of an I section loaded in the strong axis using the LRFD and ASD methods per AISC 360-16.

**References**


**Related Links**

- D1.A.5.4 Design for Shear (on page 1373)

**Details**

From the reference:

> determine the available shear strength and adequacy of an ASTM A992 W24X62 with end shears of 48 kips from dead load and 145 kips from live load.

The shear capacity values are evaluated from AISC Manual Table 2-4 and the also calculated.

**Validation**

Area of the web used for shear is:

\[
A_w = d t_w = 23.7 \times 0.430 = 10.2 \text{ in}^2
\]

The nominal shear strength is:

\[
V_n = 0.6 F_y A_w C_v = 0.6 (50)(10.2)(1.0) = 306 \text{ kips}
\]

**LRFD**

The shear strength is:

\[
\phi V_n = 1.00(306) = 306 \text{ kips}
\]

The ultimate shear demand is:
\[ Vu = 1.2 \times (48) + 1.6 \times (145) = 289.6 \text{ kips} \]

**ASD**

The allowable shear is:

\[ V_n / Q_v = 306 / 1.5 = 204 \text{ kips} \]

The shear demand is:

\[ V_a = 48 + 145 = 193 \text{ kips} \]

**Results**

**Table 539: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear capacity, ( \phi_n V_n ), Y axis (kips) LRFD</td>
<td>306</td>
<td>305.7</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Shear demand, ( V_n ), Y axis (kips) LRFD</td>
<td>289.6</td>
<td>289.6</td>
<td>none</td>
<td>The reference rounds this value to 290 kips.</td>
</tr>
<tr>
<td>Shear capacity, ( V_n / Q_v ), Y axis (kips) ASD</td>
<td>204</td>
<td>203.8</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Shear demand, ( V_a ), Y axis (kips) ASD</td>
<td>193</td>
<td>193.0</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2016\Shear\AISC 360-16 Shear Strong Axis.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Nov-18
JOB NAME AISC Design Example v15.0 - G.1
END JOB INFORMATION
*********************************************
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
*********************************************
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
```
**Verification Examples**

**V.09 Steel Design**

DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL

******************************************************************************
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X62
*
CONSTANTS
MATERIAL STEEL_50_KSI ALL
*
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
******************************************************************************
UNIT INCHES KIP
LOAD 1 LOADTYPE Dead TITLE Dead
MEMBER LOAD
1 CON GY -96
LOAD 2 LOADTYPE Live TITLE Live
MEMBER LOAD
1 CON GY -290
*
UNIT FEET KIP
LOAD COMB 3 LRFD COMBINATION
1 1.2 2 1.6
*
LOAD COMB 4 ASD Combination
1 1.0 2 1.0
******************************************************************************
PERFORM ANALYSIS
UNIT INCHES KIP
PRINT SUPPORT REACTION ALL
******************************************************************************
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
*
TRACK 2 ALL
CHECK CODE ALL
******************************************************************************
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
*
TRACK 2 ALL
CHECK CODE ALL
FINISH

---

**STAAD Output**

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
******************************************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the notations and axes labels as defined in the AISC 360-16 code. The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16 design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING:

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being \( \text{about} \) the relevant axis (X/Y), while the results for shear are reported as being for shear forces \( \text{along} \) the axis.
   E.g: Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

- F-T-B = Flexural-Torsional Buckling
- L-T-B = Lateral-Torsional Buckling
- F-L-B = Flange Local Buckling
- W-L-B = Web Local Buckling
- L-L-B = Leg Local Buckling
- C-F-Y = Compression Flange Yielding
- T-F-Y = Tension Flange Yielding

STAAD SPACE -- PAGE NO.

5

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

- Member : 1

---

STAAD.Pro 3869 User Manual
### Member Information

<table>
<thead>
<tr>
<th>Member No:</th>
<th>1</th>
<th>Profile:</th>
<th>ST W24X62</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>FAIL</td>
<td>Ratio:</td>
<td>1.009</td>
<td>Loadcase: 3</td>
</tr>
<tr>
<td>Location:</td>
<td>24.00</td>
<td>Ref:</td>
<td>Cl.F2.1</td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>0.000</td>
<td>T</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vy:</td>
<td>289.6</td>
<td>Vx:</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My:</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Mx:</td>
<td>-6950.</td>
<td></td>
</tr>
</tbody>
</table>

### SLENDERNESS

- Actual Slenderness Ratio: 34.863
- Allowable Slenderness Ratio: 300.000
- LOC: 0.00

### STRENGTH CHECKS

- Critical L/C: 3
- Ratio: 1.009 (FAIL)
- Loc: 24.00
- Condition: Cl.F2.1

### Section Properties

- (Loc: 24.00, Properties Unit: IN)
- \( A_g \): 1.820E+01
- \( A_{xx} \): 8.307E+00
- \( A_{yy} \): 1.019E+01
- \( I_{xx} \): 1.550E+03
- \( I_{yy} \): 3.450E+01
- \( J \): 1.710E+00
- \( S_{xx+} \): 1.308E+02
- \( S_{xx-} \): 1.308E+02
- \( Z_{xx} \): 1.530E+02
- \( S_{yy+} \): 9.801E+00
- \( S_{yy-} \): 9.801E+00
- \( Z_{yy} \): 1.570E+01
- \( C_w \): 4.581E+03
- \( x_0 \): 0.000E+00
- \( y_0 \): 0.000E+00

### Material Properties

- \( F_{yd} \): 50.000
- \( F_u \): 62.000

- Actual Member Length: 48.000

### Design Parameters

- \( K_x \): 1.00
- \( K_y \): 1.00
- \( NSF \): 1.00
- \( SLF \): 1.00
- \( CSP \): 12.00

### Compression Classification

- (L/C: 3, LOC: 48.00)
### Verification Examples

**V.09 Steel Design**

---

**STAAD SPACE -- PAGE NO. 6**

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE NOTED).**

- **Member:** 1 Contd.

---

**CHECKS FOR AXIAL TENSION**

### TENSILE YIELDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>819.0</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom. Ten. Yld Cap : \( P_n = 910.00 \) kip \( \text{Eq.D2-1} \)

---

### TENSILE RUPTURE

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>846.3</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Effective area : \( A_e = 18.200 \) in2 \( \text{Eq.D3-1} \)

Nom. Ten. Rpt Cap : \( P_n = 1128.4 \) kip \( \text{Eq.D2-2} \)
CHECKS FOR AXIAL COMPRESSION

FLEXURAL BUCKLING X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>709.6</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Effective Slenderness: \( L_{cx}/r_{x} = 5.2013 \) Cl.E2
- Elastic Buckling Stress: \( F_{ex} = 10580 \text{ ksi} \) Eq.E3-4
- Crit. Buckling Stress: \( F_{crx} = 49.901 \text{ ksi} \) Eq.E3-2
- Nom. Flexural Buckling: \( P_{nx} = 788.46 \text{ kip} \) Eq.E7-1

FLEXURAL BUCKLING Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>661.2</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Effective Slenderness: \( L_{cy}/r_{y} = 34.863 \) Cl.E2
- Elastic Buckling Stress: \( F_{ey} = 235.48 \text{ ksi} \) Eq.E3-4
- Crit. Buckling Stress: \( F_{cry} = 45.748 \text{ ksi} \) Eq.E3-2
- Nom. Flexural Buckling: \( P_{ny} = 734.66 \text{ kip} \) Eq.E7-1

FLEX-TOR-BUCKLING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>678.9</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Elastic F-T-B Stress: \( F_{e} = 371.19 \text{ ksi} \) Eq.E4-2
**Critical F-T-B Stress**  
\[ F_{cr} = 47.259 \text{ ksi} \]  

**Nominal Flex-tor Buckling**  
\[ P_n = 754.32 \text{ kip} \]

---

**CHECKS FOR SHEAR**

**SHEAR ALONG X**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>224.3</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results:**

- Coefficient \( C_v \) Along X: \( C_v = 1.0000 \)
- Coefficient \( K_v \) Along X: \( K_v = 1.2000 \)
- Nominal Shear Along X: \( V_{nx} = 249.22 \text{ kip} \)

**SHEAR ALONG Y**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>289.6</td>
<td>305.7</td>
<td>0.947</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results:**

- Coefficient \( C_v \) Along Y: \( C_v = 1.0000 \)
- Coefficient \( K_v \) Along Y: \( K_v = 5.3400 \)
- Nominal Shear Along Y: \( V_{ny} = 305.73 \text{ kip} \)
### CHECKS FOR BENDING

#### FLEX YIELDING (X)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>-6950.</td>
<td>6885.</td>
<td>1.009</td>
<td>Cl.F2.1</td>
<td>3</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom Flex Yielding Along X : Mnx = 7650.0 kip-in Eq.F2-1

#### FLEX. YIELDING (Y)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>705.7</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom Flex Yielding Along Y : Mny = 784.09 kip-in Eq.F6-1
| Modification Factor       : Cb     =  1.0000                Eq.H1-2 |
| Axial Capacity            : Pc     =  819.00     kip        Cl.H1.1 |
| Moment Capacity           : Mcx    =  6885.0     kip-in     Cl.H1.1 |
| Moment Capacity           : Mcy    =  705.68     kip-in     Cl.H1.1 |

---

61. *********************************************
62. LOAD LIST 4
63. PARAMETER 2
64. CODE AISC UNIFIED 2016
65. METHOD ASD
66. *
67. TRACK 2 ALL
68. CHECK CODE ALL
STEEL DESIGN
STAAD SPACE

-- PAGE NO.

STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.1)
*******************************************************************************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code. The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING: -

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/
3. Bending results are reported as being *about* the relevant axis (X/Y), while the results for shear are reported as being for shear forces *along* the axis.

E.g.: Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

---

### ABBREVIATIONS

- **F-T-B** = Flexural-Torsional Buckling
- **L-T-B** = Lateral-Torsional Buckling
- **F-L-B** = Flange Local Buckling
- **W-L-B** = Web Local Buckling
- **L-L-B** = Leg Local Buckling
- **C-F-Y** = Compression Flange Yielding
- **T-F-Y** = Tension Flange Yielding

---

**STAAD SPACE**

---

### STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.1)

---

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).**

---

<table>
<thead>
<tr>
<th>Member No:</th>
<th>1</th>
<th>Profile: ST W24X62 (AISC SECTIONS)</th>
<th>Status: FAIL</th>
<th>Ratio: 1.011</th>
<th>Loadcase: 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>24.00</td>
<td>Ref: Cl.F2.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>0.000</td>
<td>T</td>
<td>Vy: 193.0</td>
<td>Vx: 0.000</td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My: 0.000</td>
<td>Mx: -4632.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### SLENDERNESS

- Actual Slenderness Ratio : 34.863
- Allowable Slenderness Ratio : 300.000 LOC : 0.00

### STRENGTH CHECKS

- Critical L/C : 4 Ratio : 1.011(FAIL)
  - Loc : 24.00 Condition : Cl.F2.1

### SECTION PROPERTIES (LOC: 24.00, PROPERTIES UNIT: IN )

- Ag : 1.820E+01 Axx : 8.307E+00 Ayy : 1.019E+01
MATERIAL PROPERTIES

Fyld:          50.000              Fu:          62.000

Actual Member Length:        48.000

Design Parameters (Rolled)
Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00

COMPRESSON CLASSIFICATION (L/C:      4 LOC:    48.00)

<table>
<thead>
<tr>
<th>CASE</th>
<th>l</th>
<th>l p</th>
<th>l r</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange: NonSlender</td>
<td>5.97</td>
<td>N/A</td>
<td>13.49</td>
</tr>
<tr>
<td>Web</td>
<td>52.37</td>
<td>N/A</td>
<td>35.88</td>
</tr>
</tbody>
</table>

FLEXURE CLASSIFICATION (L/C:      4 LOC:    48.00)

<table>
<thead>
<tr>
<th>CASE</th>
<th>l</th>
<th>l p</th>
<th>l r</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange: Compact</td>
<td>5.97</td>
<td>9.15</td>
<td>24.08</td>
</tr>
<tr>
<td>Web</td>
<td>52.37</td>
<td>90.55</td>
<td>137.27</td>
</tr>
</tbody>
</table>

---

CHECKS FOR AXIAL TENSION

TENSILE YIELDING
### Intermediate Results:

#### Nom. Ten. Yld Cap

Nom. Ten. Yld Cap: \( P_n = 910.00 \text{ kip} \) Eq.D2-1

#### Effective area

Effective area: \( A_e = 18.200 \text{ in}^2 \) Eq.D3-1

#### Nom. Ten. Rpt Cap

Nom. Ten. Rpt Cap: \( P_n = 1128.4 \text{ kip} \) Eq.D2-2

### Checks for Axial Compression

#### Effective Slenderness

Effective Slenderness: \( L_{cx}/r_x = 5.2013 \) Cl.E2

#### Elastic Buckling Stress

Elastic Buckling Stress: \( F_{ex} = 10580. \text{ ksi} \) Eq.E3-4

#### Crit. Buckling Stress

Crit. Buckling Stress: \( F_{crx} = 49.901 \text{ ksi} \) Eq.E3-2

#### Nom. Flexural Buckling

Nom. Flexural Buckling: \( P_{nx} = 788.46 \text{ kip} \) Eq.E7-1

### Flexural Buckling Y

#### Demand

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>439.9</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Flexural Buckling X

#### Demand

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>472.1</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Tensile Rupture

#### Demand

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>564.2</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Verification Examples

STAAD.Pro 3878 User Manual
Intermediate Results:

- **Effective Slenderness**: $L_{cy}/r_y = 34.863$ (C1.E2)
- **Elastic Buckling Stress**: $F_{ey} = 235.48$ ksi (Eq.E3-4)
- **Crit. Buckling Stress**: $F_{cry} = 45.748$ ksi (Eq.E3-2)
- **Nom. Flexural Buckling**: $P_{ny} = 734.66$ kip (Eq.E7-1)

---

**FLEX-TOR-BUCKLING**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>451.7</td>
<td>0.000</td>
<td>C1.E4</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- **Elastic F-T-B Stress**: $F_e = 371.19$ ksi (Eq.E4-2)
- **Crit. F-T-B Stress**: $F_{cr} = 47.259$ ksi (Eq.E3-2)
- **Nom. Flex-tor Buckling**: $P_n = 754.32$ kip (Eq.E7-1)

---

STAAD SPACE

- **STAAD.PRO Code Checking - AISC 360-16 ASD (V1.1)**
- **Verification Examples**
  - **V.09 Steel Design**
  - **STAAD.Pro 3879 User Manual**
Nom. Shear Along X : Vnx = 249.22 kip Eq.G6-1

SHEAR ALONG Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>193.0</td>
<td>203.8</td>
<td>0.947</td>
<td>Cl.G1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Coefficient Cv Along Y : Cv = 1.0000 -
Coefficient Kv Along Y : Kv = 5.3400 Eq.G2-5
Nom. Shear Along Y : Vny = 305.73 kip Eq.G2-1

FLEX YIELDING (X)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>-4632.</td>
<td>4581.</td>
<td>1.011</td>
<td>Cl.F2.1</td>
<td>4</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom Flex Yielding Along X : Mnx = 7650.0 kip-in Eq.F2-1

FLEX. YIELDING (Y)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>469.5</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
STAAD SPACE

--- PAGE NO. ---

STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE NOTED).

- Member : 1

Contd.

CHECKS FOR AXIAL BEND INTERACTION

CLAUSE H1

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.011</td>
<td>Eq.H1-1b</td>
<td>4</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Modification Factor : \( C_b = 1.0000 \) \hspace{1cm} \( \text{Eq.H1-2} \)

Axial Capacity : \( P_c = 544.91 \) kip \hspace{1cm} \( \text{Cl.H1.1} \)

Moment Capacity : \( M_{cx} = 4580.8 \) kip-in \hspace{1cm} \( \text{Cl.H1.1} \)

Moment Capacity : \( M_{cy} = 469.52 \) kip-in \hspace{1cm} \( \text{Cl.H1.1} \)

---

V. AISC 360-16 Shear Weak Axis

Verify the shear strength of an I section loaded in the weak axis using the LRFD and ASD methods per AISC 360-16.

References


Related Links

- D1.A.5.4 Design for Shear (on page 1373)

Details

From the reference:

Verify the available shear strength and adequacy of an ASTM A992 W21X48 beam with end shears of 20.0 kips from dead load and 60.0 kips from live load in the weak direction.

Validation

---
\( C_{v2} = 1.0 \) for all ASTM A6 W shapes with \( F_y \leq 70 \) ksi, per the User Note in section G6 of the AISC 360-16 specification.

The nominal shear strength is:
\[
V_n = 2 \times [0.6 F_y b ti C_{v2}] = 2 \times [0.6 (50)(8.14)(0.43)(1.0)] = 210.0 \text{ kips}
\]

**LRFD**

The shear strength is:
\[
\phi V_n = 0.90(210.0) = 189.0 \text{ kips}
\]

The ultimate shear demand is:
\[
V_u = 1.2 (20) + 1.6 (60) = 120 \text{ kips}
\]

**ASD**

The allowable shear is:
\[
V_n / \Omega_v = 306 / 1.67 = 125.8 \text{ kips}
\]

The shear demand is:
\[
V_a = 20 + 60 = 80 \text{ kips}
\]

**Results**

**Table 540: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear capacity, ( \phi V_n ), X axis (kips) LRFD</td>
<td>189</td>
<td>189.0</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear demand, ( V_u ), X axis (kips) LRFD</td>
<td>120</td>
<td>120.0</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear capacity, ( V_n / \Omega_v ), X axis (kips) ASD</td>
<td>125.8</td>
<td>125.8</td>
<td>none</td>
<td>The reference rounds this value to 126 kips.</td>
</tr>
<tr>
<td>Shear demand, ( V_a ), X axis (kips) ASD</td>
<td>80</td>
<td>80.0</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2016\Shear\AISC 360-16 Shear Weak Axis.STD is typically installed with the program.
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
*****************************************************************************
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
*****************************************************************************
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
*
CONSTANTS
MATERIAL STEEL_50_KSI ALL
BETA 90.0 MEMBER 1
*
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
*****************************************************************************
UNIT INCHES KIP
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -40
LOAD 2 LOADTYPE Live TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -120
*
UNIT FEET KIP
LOAD COMB 3 LRFD COMBINATION
1 1.2 2 1.6
*
LOAD COMB 4 ASD Combination
1 1.0 2 1.0
*****************************************************************************
PERFORM ANALYSIS
UNIT INCHES KIP
PRINT SUPPORT REACTION ALL
*****************************************************************************
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
*
TRACK 2 ALL
CHECK CODE ALL
*****************************************************************************
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
STAAD Output

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
************************************************************************************************

ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).

***NOTE : AISC 360-16 Design Statement for STAAD.Pro.

*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the notations and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16 design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING:

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis.
   E.g : Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).

- Member : 1

<table>
<thead>
<tr>
<th>Member No:</th>
<th>1</th>
<th>Profile:</th>
<th>ST W21X48</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>FAIL</td>
<td>Ratio:</td>
<td>4.346</td>
<td>Loadcase: 3</td>
</tr>
<tr>
<td>Location:</td>
<td>24.00</td>
<td>Ref:</td>
<td>Cl.F6.2</td>
<td></td>
</tr>
</tbody>
</table>

| Pz: | 0.000 | T | Vy: | 0.000 | Vx: | -120.0 |
| Tz: | 0.000 | My: | -2880. | Mx: | 0.000 |

SLENDERNESS

Actual Slenderness Ratio : 28.973
Allowable Slenderness Ratio : 300.000 LOC : 0.00

STRENGTH CHECKS

Critical L/C : 3 Ratio : 4.346(FAIL)

| Loc : 24.00 | Condition : Cl.F6.2 |

SECTION PROPERTIES (LOC: 24.00, PROPERTIES UNIT: IN)

| Ag : 1.410E+01 | Axx : 7.000E+00 | Ayy : 7.210E+00 |
| Ixx : 9.590E+02 | Iyy : 3.870E+01 | J : 8.030E-01 |
| Sxx+ : 9.311E+01 | Sxx- : 9.311E+01 | Zxx : 1.070E+02 |
| Syy+ : 9.509E+00 | Syy- : 9.509E+00 | Zyy : 1.490E+01 |
| Cw : 3.931E+03 | x0 : 0.000E+00 | y0 : 0.000E+00 |

MATERIAL PROPERTIES

| Fyld: 50.000 | Fu: 62.000 |
**Verification Examples**

**V.09 Steel Design**

---

**Actual Member Length:** 48.000

**Design Parameters** *(Rolled)*

- $K_x$: 1.00
- $K_y$: 1.00
- NSF: 1.00
- SLF: 1.00
- CSP: 12.00

---

**COMPRESSION CLASSIFICATION (L/C: 3 LOC: 48.00)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonSlender</th>
<th>9.47</th>
<th>N/A</th>
<th>13.49</th>
<th>Table.4.1a.Case1</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Web: Slender</td>
<td>56.40</td>
<td>N/A</td>
<td>35.88</td>
<td>Table.4.1a.Case5</td>
</tr>
</tbody>
</table>

---

**FLEXURE CLASSIFICATION (L/C: 3 LOC: 48.00)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonCompact</th>
<th>9.47</th>
<th>9.15</th>
<th>24.08</th>
<th>Table.4.1b.Case13</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Web: Compact</td>
<td>56.40</td>
<td>*****</td>
<td>*****</td>
<td>Table.4.1.</td>
</tr>
</tbody>
</table>

---

**STAAD SPACE**

---

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

---

**CHECKS FOR AXIAL TENSION**

---

**TENSILE YIELDING**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>634.5</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Nom. Ten. Yld Cap: $P_n = 705.00$ kip

---

**TENSILE RUPTURE**

---
Intermediate Results:

Effective area : \( Ae = 14.100 \text{ in}^2 \) Eq.D3-1
Nom. Ten. Rpt Cap : \( Pn = 874.20 \text{ kip} \) Eq.D2-2

CHECKS FOR AXIAL COMPRESSION

FLEXURAL BUCKLING X

Intermediate Results:

Effective Slenderness : \( Lcx/rx = 5.8202 \) Cl.E2
Elastic Buckling Stress : \( Fex = 8449.2 \text{ ksi} \) Eq.E3-4
Crit. Buckling Stress : \( Fcrx = 49.876 \text{ ksi} \) Eq.E3-2
Nom. Flexural Buckling : \( Pnx = 602.71 \text{ kip} \) Eq.E7-1

FLEXURAL BUCKLING Y

Intermediate Results:

Effective Slenderness : \( Lcy/ry = 28.973 \) Cl.E2
Elastic Buckling Stress : \( Fey = 340.96 \text{ ksi} \) Eq.E3-4
Crit. Buckling Stress : \( Fcry = 47.023 \text{ ksi} \) Eq.E3-2
Nom. Flexural Buckling : \( Pny = 573.86 \text{ kip} \) Eq.E7-1
### FLEX-TOR-BUCKLING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>524.9</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
- Elastic F-T-B Stress: \( F_e = 498.48 \text{ ksi} \) (Eq.E4-2)
- Critical F-T-B Stress: \( F_{cr} = 47.944 \text{ ksi} \) (Eq.E3-2)
- Nominal Flex-tor Buckling: \( P_n = 583.20 \text{ kip} \) (Eq.E7-1)

---

### SHEAR ALONG X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>120.0</td>
<td>189.0</td>
<td>0.635</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
- Coefficient Cv Along X: \( C_v = 1.0000 \) (Eq.G2-9)
- Coefficient Kv Along X: \( K_v = 1.2000 \) (Cl.G6)
- Nominal Shear Along X: \( V_{nx} = 210.01 \text{ kip} \) (Eq.G6-1)

---

### SHEAR ALONG Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>194.7</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
<table>
<thead>
<tr>
<th>Coefficient Cv Along Y</th>
<th>Cv</th>
<th>1.0000</th>
<th>-</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coefficient Kv Along Y</td>
<td>Kv</td>
<td>5.3400</td>
<td>Eq.G2-5</td>
</tr>
<tr>
<td>Nom. Shear Along Y</td>
<td>Vny</td>
<td>216.30 kip</td>
<td>Eq.G2-1</td>
</tr>
</tbody>
</table>

---

**STAAD SPACE**

```
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

- Member : 1 Contd.
```

---

**CHECKS FOR BENDING**

**FLEX. YIELDING (Y)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>2880.</td>
<td>670.5</td>
<td>4.295</td>
<td>Cl.F6.1</td>
<td>3</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom Flex Yielding Along Y : Mny = 745.00 kip-in Eq.F6-1

---

**FLANGE LOCAL BUCK(X)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>4775.</td>
<td>0.000</td>
<td>Cl.F3.1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom F-L-B Cap : Mnx = 5306.1 kip-in Eq.F3-1

---

**FLANGE LOCAL BUCK(Y)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>2880.</td>
<td>662.7</td>
<td>4.346</td>
<td>Cl.F6.2</td>
<td>3</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom F-L-B Cap : Mny = 736.35 kip-in Eq.F6-2

-- PAGE NO.

STAAD SPACE

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

***************************

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

- Member : 1 Contd.

CHECKS FOR AXIAL BEND INTERACTION

CLAUSE H1

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.346</td>
<td>Eq.H1-1b</td>
<td>3</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Modification Factor : Cb = 1.0000 Eq.H1-2

Axial Capacity : Pc = 634.50 kip Cl.H1.1

Moment Capacity : Mcx = 4775.5 kip-in Cl.H1.1

Moment Capacity : Mcy = 662.71 kip-in Cl.H1.1

---

62. *********************************************
63. LOAD LIST 4
64. PARAMETER 2
65. CODE AISC UNIFIED 2016
66. METHOD ASD
67. *
68. TRACK 2 ALL
69. CHECK CODE ALL

STEEL DESIGN

STAAD SPACE

-- PAGE NO.

10

STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.1)

***************************

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

***NOTE : AISC 360-16 Design Statement for STAAD.Pro.

*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16 design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING:

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.

2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.

3. Bending results are reported as being ⪫ about ⪫ the relevant axis (X/Y), while the results for shear are reported as being for shear forces ⪫ along ⪫ the axis.

E.g.: Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

- F-T-B = Flexural-Torsional Buckling
- L-T-B = Lateral-Torsional Buckling
- F-L-B = Flange Local Buckling
- W-L-B = Web Local Buckling
- L-L-B = Leg Local Buckling
- C-F-Y = Compression Flange Yielding
- T-F-Y = Tension Flange Yielding

STAAD SPACE -- PAGE NO.

---

STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

- Member : 1

| Member No. | Profile: ST W21X48 (AISC SECTIONS) | Status: FAIL | Ratio: 4.354 | Loadcase: 4 | Location: 24.00 | Ref: Cl.F6.2 |
| Pz: 0.000 | T | Vy: 0.000 | Vx: -80.00 |
| Tz: 0.000 | My: -1920. | Mx: 0.000 |

**SLENDERNESS**

Actual Slenderness Ratio : 28.973
Allowable Slenderness Ratio : 300.000

**STRENGTH CHECKS**

Critical L/C : 4 Ratio : 4.354 (FAIL)
Loc : 24.00 Condition : Cl.F6.2

**SECTION PROPERTIES** (Loc: 24.00, Properties Unit: IN)

<table>
<thead>
<tr>
<th>Ag</th>
<th>Axx</th>
<th>Ayy</th>
<th>Ixx</th>
<th>Iyy</th>
<th>J</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.410E+01</td>
<td>7.000E+00</td>
<td>7.210E+00</td>
<td>9.590E+02</td>
<td>3.870E+01</td>
<td>8.030E-01</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Sxx:+</th>
<th>Sxx:-</th>
<th>Syy:+</th>
<th>Syy:-</th>
<th>Zxx</th>
<th>Zyy</th>
<th>Cw</th>
<th>x0</th>
<th>y0</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.311E+01</td>
<td>9.311E+01</td>
<td>9.509E+00</td>
<td>9.509E+00</td>
<td>1.070E+02</td>
<td>1.490E+01</td>
<td>3.931E+03</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

**MATERIAL PROPERTIES**

Fyld: 50.000 Fu: 62.000

Actual Member Length: 48.000

Design Parameters (Rolled)

Kx: 1.00 Ky: 1.00 NSF: 1.00 SLF: 1.00 CSP: 12.00

**COMPRESSION CLASSIFICATION** (L/C: 4 LOC: 48.00)

<table>
<thead>
<tr>
<th>CASE</th>
<th>l</th>
<th>l p</th>
<th>l r</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange: NonSlender</td>
<td>9.47</td>
<td>N/A</td>
<td>13.49</td>
</tr>
<tr>
<td>Web: Slender</td>
<td>56.40</td>
<td>N/A</td>
<td>35.88</td>
</tr>
</tbody>
</table>
### FLEXURE CLASSIFICATION

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonCompact</th>
<th>Web: Compact</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>9.47</td>
<td>56.40</td>
</tr>
<tr>
<td>1 p</td>
<td>9.15</td>
<td>******</td>
</tr>
<tr>
<td>1 r</td>
<td>24.08</td>
<td>******</td>
</tr>
</tbody>
</table>

Table 4.1b.Case13

Table 4.1.

---

### STAAD SPACE

**STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.1)**

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

---

#### CHECKS FOR AXIAL TENSION

**TENSILE YIELDING**

<table>
<thead>
<tr>
<th>Demanding</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>422.2</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom. Ten. Yld Cap : Pn = 705.00 kip Eq.D2-1

---

**TENSILE RUPTURE**

<table>
<thead>
<tr>
<th>Demanding</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>437.1</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Effective area : Ae = 14.100 in2 Eq.D3-1


---

**CHECKS FOR AXIAL COMPRESSION**
### FLEXURAL BUCKLING X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>360.9</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
- Effective Slenderness: \( L_{cX}/r_x = 5.8202 \) Cl.E2
- Elastic Buckling Stress: \( F_{ex} = 8449.2 \text{ ksi} \) Eq.E3-4
- Crit. Buckling Stress: \( F_{crx} = 49.876 \text{ ksi} \) Eq.E3-2
- Nom. Flexural Buckling: \( P_{nx} = 602.71 \text{ kip} \) Eq.E7-1

### FLEXURAL BUCKLING Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>343.6</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
- Effective Slenderness: \( L_{cy}/r_y = 28.973 \) Cl.E2
- Elastic Buckling Stress: \( F_{ey} = 340.96 \text{ ksi} \) Eq.E3-4
- Crit. Buckling Stress: \( F_{cry} = 47.023 \text{ ksi} \) Eq.E3-2
- Nom. Flexural Buckling: \( P_{ny} = 573.86 \text{ kip} \) Eq.E7-1

### FLEX-TOR-BUCKLING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>349.2</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
- Elastic F-T-B Stress: \( F_e = 498.48 \text{ ksi} \) Eq.E4-2
- Crit. F-T-B Stress: \( F_{cr} = 47.944 \text{ ksi} \) Eq.E3-2
- Nom. Flex-tor Buckling: \( P_n = 583.20 \text{ kip} \) Eq.E7-1
CHECKS FOR SHEAR

SHEAR ALONG X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>80.00</td>
<td>125.8</td>
<td>0.636</td>
<td>Cl.G1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Coefficient Cv Along X : Cv = 1.0000
Coefficient Kv Along X : Kv = 1.2000
Nom. Shear Along X : Vnx = 210.01 kip

SHEAR ALONG Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>129.5</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Coefficient Cv Along Y : Cv = 1.0000
Coefficient Kv Along Y : Kv = 5.3400
Nom. Shear Along Y : Vny = 216.30 kip

CHECKS FOR BENDING
### FLEX. YIELDING (Y)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>1920.</td>
<td>446.1</td>
<td>4.304</td>
<td>Cl.F6.1</td>
<td>4</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom Flex Yielding Along Y: \( M_{ny} = 745.00 \) kip-in \( \text{Eq.F6-1} \)

### FLANGE LOCAL BUCK(X)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>3177.</td>
<td>0.000</td>
<td>Cl.F3.1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom F-L-B Cap: \( M_{nx} = 5306.1 \) kip-in \( \text{Eq.F3-1} \)

### FLANGE LOCAL BUCK(Y)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>1920.</td>
<td>440.9</td>
<td>4.354</td>
<td>Cl.F6.2</td>
<td>4</td>
<td>24.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom F-L-B Cap: \( M_{ny} = 736.35 \) kip-in \( \text{Eq.F6-2} \)
Intermediate Results:

- Modification Factor: $C_b = 1.0000$ Eq.H1-2
- Axial Capacity: $P_c = 422.16$ kip Cl.H1.1
- Moment Capacity: $M_{cx} = 3177.3$ kip-in Cl.H1.1
- Moment Capacity: $M_{cy} = 440.93$ kip-in Cl.H1.1

**V. Tension**

V. AISC 360-16 I Tension LRFD

To verify the tensile yield strength of an I section using the LRFD method from the AISC 360-16 code.

**Reference**


**Details**

Verify the tensile yield and tensile rupture strength of an I section (W8X21) using the LRFD method. The material is A992 ($F_y = 50$ ksi, $F_u = 65$ ksi)

The area of the bolt holes is assumed to be zero. A known shear lag factor of 0.908 is directly specified.

The length of the member is 25 ft. It is loaded axial in tension with a dead load of 30 kips and a live load of 90 kips.

**Validation**

The factored axial load, $P_u = 1.2 \times 30 + 1.6 \times 90 = 180$ kips

**Tensile Yield Strength**

The tensile yield strength: $\phi_{tY}P_{nY} = \phi_{tY} \times F_y \times A_g = 0.9 \times 50 \times 6.16 = 277.2$ kips

Ratio for yield strength: 0.649

**Tensile Rupture Strength**

For this example, $A_n$ is assumed equal to $A_g$.

Effective area, $A_e = A_n \times U = 6.16 \times 0.908 = 5.593$ in$^2$

The tensile rupture strength = $\phi_{tR}P_{nR} = \phi_{tR} \times F_u \times A_e = 0.75 \times 65 \times 5.593 = 272.672$ kips

Ratio for rupture strength: 0.660

**Results**
### Table 541: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile yield strength (kips)</td>
<td>277.2</td>
<td>277.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Ratio</td>
<td>0.649</td>
<td>0.649</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Tensile rupture strength (kips)</td>
<td>272.7</td>
<td>272.7</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Ratio</td>
<td>0.660</td>
<td>0.660</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2016\Tension\AISC 360-16 I Tension LRFD.STD is typically installed with the program.

```plaintext
STAAD SPACE
START JOB INFORMATION
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 9360 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X21
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
JOINT LOAD
2 FX 30
LOAD 2 LOADTYPE Live TITLE LOAD CASE 2
JOINT LOAD
2 FX 90
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
```

STAAD.Pro 3898 User Manual
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FU 9360 ALL
FYLD 7200 ALL
KX 1 ALL
KY 1 ALL
KZ 1 ALL
METHOD LRFD
PROFILE W8X ALL
*SGR 28 ALL
SLF 0.908 ALL
STP 1 ALL
TRACK 2 ALL
UNIT KIP INCH
CHECK CODE ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
**********************************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the
notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING: -

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that
produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase
Based on the code clause in AISC 360-16 will not be shown in the report.

3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis.

E.g: Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
F-L-B = Flange Local Buckling
W-L-B = Web Local Buckling
L-L-B = Leg Local Buckling
C-F-Y = Compression Flange Yielding
T-F-Y = Tension Flange Yielding

STAAD SPACE

--- PAGE NO. 4 ---

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

- Member :  1

<table>
<thead>
<tr>
<th>Member No:</th>
<th>1</th>
<th>Profile:</th>
<th>ST W8X21 (AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>PASS</td>
<td>Ratio:</td>
<td>0.660</td>
</tr>
<tr>
<td>Location:</td>
<td>0.00</td>
<td>Ref:</td>
<td>Cl.D2</td>
</tr>
<tr>
<td>Pz:</td>
<td>180.0</td>
<td>T</td>
<td>Vx:</td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My:</td>
<td>0.000</td>
</tr>
</tbody>
</table>

--- SLENDERNESS ---

Actual Slenderness Ratio : 238.212
Allowable Slenderness Ratio : 300.000 LOC : 0.00

--- STRENGTH CHECKS ---

Critical L/C : 3 Ratio : 0.660(PASS)

Loc : 0.00 Condition : Cl.D2

--- SECTION PROPERTIES (LOC: 0.00, PROPERTIES UNIT: IN ) ---

Ag : 6.160E+00 Axx : 4.216E+00 Ayy : 2.070E+00
Ixx : 7.530E+01  Iyy : 9.770E+00  J : 2.820E-01
Sxx+: 1.819E+01  Sxx-: 1.819E+01  Zxx : 2.040E+01
Syy+: 3.708E+00  Syy-: 3.708E+00  Zyy : 5.690E+00
Cw : 1.515E+02  x0 : 0.000E+00  y0 : 0.000E+00

**MATERIAL PROPERTIES**
Fyld: 50.000  Fu: 65.000

---

**Actual Member Length:** 300.000

**Design Parameters:** (Rolled)
Kx: 1.00  Ky: 1.00  NSF: 1.00  SLF: 0.91  CSP: 12.00

---

**COMPRESSION CLASSIFICATION (L/C: 3 LOC: 300.00)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange:</th>
<th>l</th>
<th>lp</th>
<th>lr</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>NonSlender</td>
<td>6.59</td>
<td>N/A</td>
<td>13.49</td>
</tr>
<tr>
<td></td>
<td>Web</td>
<td>29.92</td>
<td>N/A</td>
<td>35.88</td>
</tr>
</tbody>
</table>

---

**FLEXURE CLASSIFICATION (L/C: 3 LOC: 300.00)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange:</th>
<th>l</th>
<th>lp</th>
<th>lr</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Compact</td>
<td>6.59</td>
<td>9.15</td>
<td>24.08</td>
</tr>
<tr>
<td></td>
<td>Web</td>
<td>29.92</td>
<td>90.55</td>
<td>137.27</td>
</tr>
</tbody>
</table>

---

**STAAD SPACE**

---

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

---

**CHECKS FOR AXIAL TENSION**

---

**TENSILE YIELDING**
<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>180.0</td>
<td>277.2</td>
<td>0.649</td>
<td>C1.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom. Ten. Yld Cap : Pn = 308.00 kip Eq.D2-1

**TENSILE RUPTURE**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>180.0</td>
<td>272.7</td>
<td>0.660</td>
<td>C1.D2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Effective area : Ae = 5.5933 in² Eq.D3-1

**CHECKS FOR AXIAL COMPRESSION**

**FLEXURAL BUCKLING X**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>161.8</td>
<td>0.000</td>
<td>C1.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Effective Slenderness : Lcx/rx = 85.805 C1.E2
Elastic Buckling Stress : Fex = 38.875 ksi Eq.E3-4
Crit. Buckling Stress : Fcrx = 29.186 ksi Eq.E3-2
Nom. Flexural Buckling : Pnx = 179.79 kip Eq.E3-1

**FLEXURAL BUCKLING Y**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>24.52</td>
<td>0.000</td>
<td>C1.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>
Intermediate Results:

- Effective Slenderness: $L_{cy}/r_y = 238.21$ (Cl.E2)
- Elastic Buckling Stress: $F_{ey} = 5,043.9$ ksi (Eq.E3-4)
- Critical Buckling Stress: $F_{cry} = 4,423.5$ ksi (Eq.E3-3)
- Nominal Flexural Buckling: $P_{ny} = 27,249$ kip (Eq.E3-1)

FLEX-TOR-BUCKLING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>169.7</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Elastic F-T-B Stress: $F_e = 42.637$ ksi (Eq.E4-2)
- Critical F-T-B Stress: $F_{cr} = 30.606$ ksi (Eq.E3-2)
- Nominal Flex-tor Buckling: $P_n = 188.53$ kip (Eq.E4-1)

STAAD SPACE

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

CHECKS FOR SHEAR

SHEAR ALONG X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>113.8</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Coefficient $C_v$ Along X: $C_v = 1.0000$ (Eq.G2-9)
- Coefficient $K_v$ Along X: $K_v = 1.2000$ (Cl.G6)
### Nom. Shear Along X

<table>
<thead>
<tr>
<th>Eq.</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>G6-1</td>
<td>126.48</td>
<td>kip</td>
</tr>
</tbody>
</table>

### SHEAR ALONG Y

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>62.10</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results:**

- Coefficient Cv Along Y: \( \text{Cv} = 1.0000 \)
- Coefficient Kv Along Y: \( \text{Kv} = 5.3400 \)
- Nom. Shear Along Y: \( \text{Vny} = 62.100 \) kip

### FLEX YIELDING (X)

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>918.0</td>
<td>0.000</td>
<td>Cl.F2.1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results:**

- Nom Flex Yielding Along X: \( \text{Mnx} = 1020.0 \) kip-in

### FLEX. YIELDING (Y)

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>256.1</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results:**
Nom Flex Yielding Along Y : Mny = 284.50 kip-in Eq.F6-1

LAT TOR BUCK ABOUT X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>302.2</td>
<td>0.000</td>
<td>Cl.F2.2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom L-T-B Cap : Mnx = 335.72 kip-in Eq.F2-3
Mom. Distr. factor : CbX = 1.0000 Eq.F1-1
Limiting Unbraced Length : LpX = 53.381 in Eq.F2-5
coefficient C : Cx = 1.0000 Eq.F2-8a
Effective Rad. of Gyr. : Rts = 1.4543 in Eq.F2-7
Limiting Unbraced Length : LrX = 176.96 in Eq.F2-6
Crit. Elas. L-T-B Stress : FcrX = 18.458 ksi Eq.F2-4

STAAD SPACE -- PAGE NO.
8

CLAUSE H1

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.660</td>
<td>Eq.H1-1a</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Modification Factor : Cb = 2.6064 Eq.H1-2
Axial Capacity : Pc = 272.67 kip Cl.H1.1
Moment Capacity : Mcx = 787.52 kip-in Cl.H1.1

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).
V. Torsion

V. AISC 360-16 - Torsion

Verify the torsional strength of an angle section using the LRFD method in the AISC 360-16 code. Use the simple method for calculating torsional shears (i.e., no full twist analysis). Perform torsion checks per AISC Design Guide 9.

Reference
2. Torsional Analysis of Structural Steel Members Design Guide 9, American Institute of Steel Construction

Details

The angle section is an L 3x3x1/2 angle of ASTM A992 material. The member is a 2 ft long cantilever with a total end torque of 3.0 in·kips.

Validation

Maximum torsional shear stress: \( \tau_f = \frac{Tt_f}{J} = 3.0 \frac{0.5}{0.234} = 6.41 \text{ksi} \)

By observation, direct shear stress is zero (torsion loading only). Therefore, total torsional stress is = \( \tau_f \).

Torsional shear resistance: \( \phi T \times 0.6 \times F_y = 0.9 \times 0.6 \times 50 = 27 \text{ksi} \)

Ratio: \( 6.41 \text{ ksi} / 27 \text{ ksi} = 0.237 \)

Results

Table 542: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Torsional shear stress</td>
<td>0.237</td>
<td>0.242</td>
<td>2.1%</td>
<td>Negligible difference when considering simplified method used by hand calculations</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2016\Torsion\AISC 360-16 - Torsion.STD is typically installed with the program.

STAAD SPACE

START JOB INFORMATION
ENGINEER DATE 23-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 2 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L30308
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CMOM GX -0.25 2
PERFORM ANALYSIS
PARAMETER 1
CODE AISC UNIFIED 2016
ALH 1 ALL
FU 7200 ALL
FYLD 7200 ALL
METHOD LRFD
SOE 0 ALL
TND 9 ALL
TORSION 1 ALL
TRACK 2 ALL
UNIT KIP INCH
CHECK CODE ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
***************************************************************************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:
========================================================================
The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
design results are reported in AISC 360-16 code axis convention.
AISC Spec. STAAD.Pro Description
---------- ---------- ------------
X Z Axis typically parallel to the sections
principal major axis.
Y Y Axis typically parallel to the sections
principal minor axis.
Z               X          Longitudinal axis perpendicular to the
cross section.
SECTION FORCES AXIS MAPPING: -
<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis.
   E.g.: Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:
F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
F-L-B = Flange Local Buckling
W-L-B = Web Local Buckling
L-L-B = Leg Local Buckling
C-F-Y = Compression Flange Yielding
T-F-Y = Tension Flange Yielding

STAAD SPACE  -- PAGE NO. 4

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).
- Member : 1

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile: ST L30308 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.242</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>0.00</td>
<td>Ref: DG9:Eq: 4.13</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>0.000</td>
<td>T Vy: 0.000 Vx: 0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>3.000</td>
<td>My: 0.000 Mx: 0.000</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
SLENDERNESS

Actual Slenderness Ratio : 41.379
Allowable Slenderness Ratio : 300.000 LOC : 0.00

STRENGTH CHECKS

Critical L/C : 1 Ratio : 0.242 (PASS)
Loc : 0.00 Condition : DG9:Eq: 4.13

SECTION PROPERTIES (LOC: 0.00, PROPERTIES UNIT: IN )

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>2.760E+00</td>
</tr>
<tr>
<td>Axx</td>
<td>1.500E+00</td>
</tr>
<tr>
<td>Ayy</td>
<td>1.500E+00</td>
</tr>
<tr>
<td>Ixx</td>
<td>2.216E+00</td>
</tr>
<tr>
<td>Iyy</td>
<td>2.216E+00</td>
</tr>
<tr>
<td>J</td>
<td>2.300E-01</td>
</tr>
<tr>
<td>Sxx+</td>
<td>7.071E-01</td>
</tr>
<tr>
<td>Sxx-</td>
<td>1.149E+00</td>
</tr>
<tr>
<td>Zxx</td>
<td>1.273E+00</td>
</tr>
<tr>
<td>Syy+</td>
<td>1.652E+00</td>
</tr>
<tr>
<td>Syy-</td>
<td>1.652E+00</td>
</tr>
<tr>
<td>Zyy</td>
<td>2.974E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>1.444E-01</td>
</tr>
<tr>
<td>x0</td>
<td>-6.784E-01</td>
</tr>
<tr>
<td>y0</td>
<td>-6.784E-01</td>
</tr>
</tbody>
</table>

MATERIAL PROPERTIES

Fyld: 50.000 Fu: 50.000

Actual Member Length: 24.000

Design Parameters (Rolled)

Kx: 1.00 Ky: 1.00 NSF: 1.00 SLF: 1.00 CSP: 12.00

COMPRESSION CLASSIFICATION (L/C: 1 LOC: 24.00)

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonSlender</th>
<th>Web : NonSlender</th>
</tr>
</thead>
<tbody>
<tr>
<td>l</td>
<td>6.00</td>
<td>6.00</td>
</tr>
<tr>
<td>lp</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>lr</td>
<td>10.84</td>
<td>10.84</td>
</tr>
<tr>
<td></td>
<td>Table.4.1a.Case3</td>
<td>Table.4.1a.Case3</td>
</tr>
</tbody>
</table>

FLEXURE CLASSIFICATION (L/C: 1 LOC: 24.00)

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonSlender</th>
<th>Web : NonSlender</th>
</tr>
</thead>
<tbody>
<tr>
<td>l</td>
<td>6.00</td>
<td>6.00</td>
</tr>
<tr>
<td>lp</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>lr</td>
<td>10.84</td>
<td>10.84</td>
</tr>
</tbody>
</table>
**STAAD SPACE**

--- PAGE NO.

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).

--- Member : 1 Contd.

### CHECKS FOR AXIAL TENSION

#### TENSILE YIELDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>124.2</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Nom. Ten. Yld Cap : Pn = 138.00 kip  Eq.D2-1

#### TENSILE RUPTURE

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>103.5</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

Effective area : Ae = 2.7600 in2  Eq.D3-1


### CHECKS FOR AXIAL COMPRESSION

#### FLEXURAL BUCKLING X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>117.9</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>
Intermediate Results:

**Effective Slenderness**: $L_{cx}/r_x = 26.782$  
*Cl. E2*

**Elastic Buckling Stress**: $F_{ex} = 399.04$ ksi  
*Eq. E3-4*

**Crit. Buckling Stress**: $F_{crx} = 47.445$ ksi  
*Eq. E3-2*

**Nom. Flexural Buckling**: $P_{nx} = 130.95$ kip  
*Eq. E3-1*

---

**FLEXURAL BUCKLING Y**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>117.9</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

**Effective Slenderness**: $L_{cy}/r_y = 26.782$  
*Cl. E2*

**Elastic Buckling Stress**: $F_{ey} = 399.04$ ksi  
*Eq. E3-4*

**Crit. Buckling Stress**: $F_{cry} = 47.445$ ksi  
*Eq. E3-2*

**Nom. Flexural Buckling**: $P_{ny} = 130.95$ kip  
*Eq. E3-1*

---

**FLEXURAL BUCKLING U**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>120.1</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

**Effective Slenderness**: $L_{cu}/r_u = 21.299$  
*Cl. E2*

**Elastic Buckling Stress**: $F_{eu} = 630.92$ ksi  
*Eq. E3-4*

**Crit. Buckling Stress**: $F_{cru} = 48.369$ ksi  
*Eq. E3-2*

**Nom. Flexural Buckling**: $P_{nu} = 133.50$ kip  
*Eq. E3-1*

---

**FLEXURAL BUCKLING V**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
</table>
Intermediate Results:

**Effective Slenderness**: \( \frac{L_{cv}}{r_{v}} = 41.379 \)  
**Elastic Buckling Stress**: \( F_{ev} = 167.16 \text{ ksi} \)  
**Critical Buckling Stress**: \( F_{crv} = 44.116 \text{ ksi} \)  
**Nom. Flexural Buckling**: \( P_{nv} = 121.76 \text{ kip} \)

---

### CHECKS FOR SHEAR

**SHEAR ALONG X**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>40.50</td>
<td>0.000</td>
<td>C1.G1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- **Coefficient Cv Along X** : \( C_{v} = 1.0000 \)  
- **Coefficient Kv Along X** : \( K_{v} = 1.2000 \)  
- **Nom. Shear Along X** : \( V_{nx} = 45.000 \text{ kip} \)

**SHEAR ALONG Y**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>40.50</td>
<td>0.000</td>
<td>C1.G1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:

- **Coefficient Cv Along Y** : \( C_{v} = 1.0000 \)
| Coefficient Kv Along Y : Kv   =  1.2000   Cl.G3 |
| Nom. Shear Along Y : Vny    =  45.000     kip   Eq.G3-1 |

| STAAD SPACE                  -- PAGE NO. 7 |

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted).**

--- Member : 1 Contd.

**CHECKS FOR BENDING**

**FLEX YIELDING (X)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>72.22</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom Flex Yielding Along X : Mnx = 80.244 kip-in Eq.F10-1

**FLEX. YIELDING (Y)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>72.22</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom Flex Yielding Along Y : Mny = 80.244 kip-in Eq.F10-1

**FLEX. YIELDING (U)**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>111.5</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom Flex Yielding Along U : Mnu = 123.90 kip-in Eq.F10-1
FLEX. YIELDING (V)

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>47.73</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom Flex Yielding Along V : Mnv = 53.034 kip-in Eq.F10-1

LAT TOR BUCK ABOUT X

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>57.78</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom L-T-B Cap : Mnx = 64.196 kip-in Eq.F10-2
Mom. Distr. factor : CbX = 1.0000 Eq.F1-1
Crit. Elastic L-T-B Mom. : McrX = 3076.5 kip-in Eq.F10-5b

LAT TOR BUCK ABOUT Y

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>57.78</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Intermediate Results:
Nom Flex Yielding About Y : Mny = 64.196 kip-in Eq.F10-2
Mod. factor Cb : CbY = 1.0000 Eq.F1-1
Crit. Elas. L-T-B Moment : McrY = 3076.5 kip-in Eq.F10-5b

LAT TOR BUCK ABOUT U

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>111.5</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>
### Intermediate Results:

- **Nom Flex Yielding About U**: \( M_{nu} = 123.90 \text{ kip-in} \)  
  Eq. F10-2
- **Modification factor Cb**: \( C_{bU} = 1.0000 \)  
  Eq. F1-1
- **Crit. Elast. L-T-B Moment**: \( M_{crU} = 1088.0 \text{ kip-in} \)  
  Eq. F10-4

---

### Checks for Axial Bend Interaction

#### Clause H2

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>Eq.H2-1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

### Normal Stress Ratio

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Intermediate Results:**

- **Torque:** \( Tq = 3.0000 \) kip-in
- **Mod. Factor:** \( \Phi = 1.0000 \)

**SHEAR STRESS RATIO**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.242</td>
<td>DG9:Eq: 4.13</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**COMB. BUCK. RATIO**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>DG9:Eq: 4.16(b)</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**V. AISC 360-2010**

**V. AISC 360-10 - E-2**

Reference


**Note:** This reference is freely available for download from the AISC website at [https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf](https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf)

**Problem**

From the reference:

Verify that a built-up, ASTM A572 Grade 50 column with PL 1 in. × 8 in. flanges and a PL 4 in. × 15 in. web is sufficient to carry a dead load of 70 kips and live load of 210 kips in axial compression. The column length is 15 ft and the ends are pinned in both axes.

Refer to the reference for calculations.

**Comparison**
Table 543: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi_c P_n$ (kips) [LRFD]</td>
<td>508</td>
<td>513</td>
<td>1.0%</td>
</tr>
<tr>
<td>$P_n/\Omega_c$ (kips) [ASD]</td>
<td>338</td>
<td>341</td>
<td>0.9%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - E-2.STD is typically installed with the program.

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.
1 PINNED
2 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
JOINT LOAD
2 FY -70
LOAD 2 LOADTYPE Live TITLE LOAD CASE 2
JOINT LOAD
2 FY -210
UNIT INCHES KIP
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 65 ALL
FYLD 50 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 65 ALL
FYLD 50 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

1

1. STAAD PLANE
INPUT FILE: AISC 360-10 - E-2.STD
2. START JOB INFORMATION
3. ENGINEER DATE 31-OCT-12
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT FEET KIP
7. JOINT COORDINATES
8. 1 0 0 0; 2 0 15 0
9. MEMBER INCIDENCES
10. 112
11. START USER TABLE
12. *TABLE 1
13. *UNIT INCHES KIP
14. *ISECTION
15. *I1
16. *17 0.25 17 8 1 8 1 0 0 0
17. TABLE 2
18. UNIT INCHES KIP
19. WIDE FLANGE
20. I
21. 19.75 17 0.25 8 1 1095.65 85.3529 5.41146 4.25 10.6667 8 1
22. END
23. UNIT INCH KIP
24. DEFINE MATERIAL START
25. ISOTROPIC STEEL
26. E 29732.7
27. POISSON 0.3
28. DENSITY 0.000283
29. ALPHA 1.2E-005
30. DAMP 0.03
31. TYPE STEEL
32. STRENGTH FY 50 FU 65 RY 1.5 RT 1.2
33. END DEFINE MATERIAL
34. UNIT INCHES KIP
35. CONSTANTS
36. MATERIAL STEEL ALL
37. MEMBER PROPERTY AMERICAN
38. 1 UPTABLE 2 I
39. STAAD PLANE
2
40. UNIT FEET KIP
41. SUPPORTS
42. 1 PINNED
43. 2 FIXED BUT FY MX MY MZ
44. LOAD 1 LOADTYPE DEAD TITLE LOAD CASE 1
45. JOINT LOAD
46. 2 FY -70
47. LOAD 2 LOADTYPE LIVE TITLE LOAD CASE 2
48. JOINT LOAD
49. 2 FY -210
50. UNIT INCHES KIP
51. LOAD COMB 3 COMBINATION LOAD CASE 3
52. 1 1.2 2 1.6
53. LOAD COMB 4 COMBINATION LOAD CASE 4
54. 1 1.0 2 1.0
55. PERFORM ANALYSIS

PROBLEM STATISTICS

--------------------------------------------------------
NUMBER OF JOINTS 2 NUMBER OF MEMBERS 1
NUMBER OF PLATES 0 NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0 NUMBER OF SUPPORTS 2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 3
TOTAL LOAD COMBINATION CASES = 2 SO FAR.
55. LOAD LIST 3
56. PARAMETER 1
## Steel Design

### Verification Examples

**V.09 Steel Design**

### STEEL DESIGN

**STAAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a**

---

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1 ST I</td>
<td>(UPT)</td>
<td>PASS</td>
<td>Eq. H1-1a</td>
<td>0.819</td>
<td>3</td>
<td>420.00 C</td>
</tr>
</tbody>
</table>

---

### SLENDERNESS

**Actual Slenderness Ratio:** 86.586  **L/C:** 3

**Allowable Slenderness Ratio:** 200.000  **LOC:** 0.00

---

### STRENGTH CHECKS

**Critical L/C:** 3  **Ratio:** 0.819 (PASS)

**Loc:** 0.00  **Condition:** Eq. H1-1a

---

### DESIGN FORCES

**Fx:** 4.200E+02 (C)  **Fy:** 0.000E+00  **Fz:** 0.000E+00

**Mx:** 0.000E+00  **My:** 0.000E+00  **Mz:** 0.000E+00

---

### SECTION PROPERTIES (UNIT: INCH)

**Azz:** 1.067E+01  **Ayy:** 4.250E+00  **Cw:** 5.461E+03

**Szz:** 1.289E+02  **Syy:** 2.134E+01

**Izz:** 1.096E+03  **Iyy:** 8.535E+01  **Ix:** 5.411E+00

---

### MATERIAL PROPERTIES

**Fyld:** 50.000  **Fu:** 65.000

---

### Actual Member Length: 180.000

---

### Design Parameters

**Kz:** 1.00  **Ky:** 1.00  **NSF:** 1.00  **SLF:** 1.00  **CSP:** 12.00

---

### SECTION CLASS UNSTIFFENED / l l p l r CASE

**STIFFENED**

**Compression:** Non-Slender 4.00  **N/A**  13.66  **T.B4.1(a)-1**

**Slender:** 60.00  **N/A**  36.33  **T.B4.1(a)-5**

**Flexure:** Compact 4.00  91.69  **T.B4.1(b)-15**

**Compact:** 60.00  **T.B4.1(b)-10**

---

**STAAD PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a**

---

**CHECK FOR AXIAL TENSION**
<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>8.89E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>3</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>9.63E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>3</td>
</tr>
</tbody>
</table>

CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>4.20E+02</td>
<td>8.03E+02</td>
<td>0.523</td>
<td>Eq. E7-1</td>
<td>3</td>
</tr>
<tr>
<td>Min Buck</td>
<td>4.20E+02</td>
<td>5.13E+02</td>
<td>0.819</td>
<td>Eq. E7-1</td>
<td>3</td>
</tr>
<tr>
<td>Flexural</td>
<td>Tor Buck</td>
<td>4.20E+02</td>
<td>6.84E+02</td>
<td>0.614</td>
<td>Eq. E7-1</td>
</tr>
</tbody>
</table>

Intermediate

Results

<table>
<thead>
<tr>
<th>Eff Area</th>
<th>KL/r</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>1.86E+01</td>
<td>24.17</td>
<td>4.52E+01</td>
<td>5.02E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>1.91E+01</td>
<td>86.59</td>
<td>2.89E+01</td>
<td>3.91E+01</td>
</tr>
<tr>
<td>Flexural</td>
<td>Tor Buck</td>
<td>1.88E+01</td>
<td>3.85E+01</td>
<td>7.60E+02</td>
</tr>
</tbody>
</table>

CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>2.88E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>1.15E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate

Results

<table>
<thead>
<tr>
<th>Aw</th>
<th>Cv</th>
<th>Kv</th>
<th>h/tw</th>
<th>Vn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>1.07E+01</td>
<td>1.00</td>
<td>1.20</td>
<td>4.00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.25E+00</td>
<td>1.00</td>
<td>5.00</td>
<td>60.00</td>
</tr>
</tbody>
</table>

CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>6.39E+03</td>
<td>0.000</td>
<td>Eq. F2-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>1.45E+03</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate

Results

<table>
<thead>
<tr>
<th>Mn</th>
<th>My</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>7.10E+03</td>
</tr>
<tr>
<td>Minor</td>
<td>1.61E+03</td>
</tr>
</tbody>
</table>

CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>5.44E+03</td>
<td>0.000</td>
<td>Eq. F2-2</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
</tr>
<tr>
<td>Major</td>
<td>6.04E+03</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>89.22</td>
<td>311.02</td>
</tr>
</tbody>
</table>

STAAD PLANE

--- PAGE NO. 5

STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a

*******************************************************************************

ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)

CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.819</td>
<td>Eq. H1-1a</td>
<td>3</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.000</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate

<table>
<thead>
<tr>
<th>Mcx /</th>
<th>Mrx /</th>
<th>Pc /</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mcy</td>
<td>Mry</td>
<td>Pr</td>
</tr>
<tr>
<td>Flexure Comp</td>
<td>5.44E+03</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>1.45E+03</td>
<td>0.00E+00</td>
<td>4.20E+02</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>5.44E+03</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>
63. LOAD LIST 4
64. PARAMETER 2
65. CODE AISC UNIFIED 2010
66. FU 65 ALL
67. FYLD 50 ALL
68. METHOD ASD
69. TRACK 2 ALL
70. CHECK CODE ALL

STEEL DESIGN

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/CRITICAL COND/LOADING/LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
</tr>
<tr>
<td>1 ST</td>
<td>I</td>
<td>PASS</td>
</tr>
<tr>
<td></td>
<td>(UPT)</td>
<td>280.00 C</td>
</tr>
</tbody>
</table>

SLENDERNESS
Actual Slenderness Ratio : 86.586
Allowable Slenderness Ratio : 200.000

STRENGTH CHECKS
Critical L/C : 4 Ratio : 0.820(PASS)
Loc : 0.00 Condition : Eq. H1-1a

DESIGN FORCES
Fx: 2.800E+02(C) Fy: 0.000E+00 Fz: 0.000E+00
Mx: 0.000E+00 My: 0.000E+00 Mz: 0.000E+00

SECTION PROPERTIES (UNIT: INCH)
Azz: 1.067E+01 Ayy: 4.250E+00 Cw: 5.461E+03
Szz: 1.289E+02 Syy: 2.134E+01
Izz: 1.096E+03 Iyy: 8.535E+01 Ix: 5.411E+00

MATERIAL PROPERTIES
Fyld: 50.000 Fu: 65.000

Actual Member Length: 180.000

Design Parameters
Kz: 1.00 Ky: 1.00 NSF: 1.00 SLF: 1.00 CSP: 12.00

SECTION CLASS UNSTIFFENED / STIFFENED

<table>
<thead>
<tr>
<th>CASE</th>
<th>l</th>
<th>lp</th>
<th>l r</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

STAAD PLANE
<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>5.91E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>6.42E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>2.80E+02</td>
<td>5.35E+02</td>
<td>0.524</td>
<td>Eq. E7-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Min Buck</td>
<td>2.80E+02</td>
<td>3.41E+02</td>
<td>0.820</td>
<td>Eq. E7-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Flexural Tor</td>
<td>2.80E+02</td>
<td>4.55E+02</td>
<td>0.615</td>
<td>Eq. E7-1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Eff Area</th>
<th>KL/r</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>1.86E+01</td>
<td>24.17</td>
<td>4.52E+01</td>
<td>5.02E+02</td>
<td>8.93E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>1.91E+01</td>
<td>86.59</td>
<td>2.89E+01</td>
<td>3.91E+01</td>
<td>5.70E+02</td>
</tr>
<tr>
<td>Flexural Tor</td>
<td>1.88E+01</td>
<td>3.85E+01</td>
<td>7.60E+02</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>1.92E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>7.63E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Aw</th>
<th>Cv</th>
<th>Kv</th>
<th>h/tw</th>
<th>Vn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>1.07E+01</td>
<td>1.00</td>
<td>1.20</td>
<td>4.00</td>
<td>3.20E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.25E+00</td>
<td>1.00</td>
<td>5.00</td>
<td>60.00</td>
<td>1.27E+02</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>4.25E+03</td>
<td>0.000</td>
<td>Eq. F2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>9.65E+02</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Mn</th>
<th>My</th>
<th>Mcx</th>
<th>Mrx</th>
<th>Pc</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>7.10E+03</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>1.61E+03</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>3.62E+03</td>
<td>0.000</td>
<td>Eq. F2-2</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
<td>Lb</td>
</tr>
<tr>
<td>Major</td>
<td>6.04E+03</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>89.22</td>
<td>311.02</td>
<td>180.00</td>
</tr>
</tbody>
</table>

**Verification Examples**

**STAAD.Pro**

**User Manual**
Verification Examples

V.09 Steel Design

### V. AISC 360-10 - G-6

**Reference**


**Note:** This reference is freely available for download from the AISC website at [https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf](https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf)

**Problem**

From the reference:

> Verify the available shear strength and adequacy of a W21×48 ASTM A992 beam with end shears of 20.0 kips from dead load and 60.0 kips from live load in the weak direction.

Refer to the reference for calculations.

**Comparison**

**Table 544: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi \psi_{n}$ [kips] [LRFD]</td>
<td>189</td>
<td>189</td>
<td>none</td>
</tr>
<tr>
<td>$V_n / \Omega_1$ [kips] [ASD]</td>
<td>126</td>
<td>126</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

### 71. FINISH

*********** END OF THE STAAD.Pro RUN ***********

**** DATE= APR 14, 2019 TIME= 23:11:15 ****

STAAD PLANE -- PAGE NO.

<table>
<thead>
<tr>
<th>Flexure Comp</th>
<th>Mcy</th>
<th>Mry</th>
<th>Pr</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.62E+03</td>
<td>0.00E+00</td>
<td>3.41E+02</td>
<td></td>
</tr>
<tr>
<td>9.65E+02</td>
<td>0.00E+00</td>
<td>2.80E+02</td>
<td></td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>3.62E+03</td>
<td>0.00E+00</td>
<td>5.91E+02</td>
</tr>
<tr>
<td>9.65E+02</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td></td>
</tr>
</tbody>
</table>
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - G-6.STD is typically installed with the program.

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

```plaintext
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 8 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.28151e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 9360 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GZ -20 0.665
LOAD 2 LOADTYPE Live TITLE LOAD CASE 2
MEMBER LOAD
1 CON GZ -60 0.665
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 7200
FU 9360
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 7200
FU 9360
TRACK 2 ALL
```
### STAAD Output

**STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a**

----------

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOADING/</td>
<td>LOCATION</td>
</tr>
<tr>
<td>---------</td>
<td>-------</td>
<td>---------</td>
<td>----------------</td>
<td>--------</td>
<td>----------</td>
<td>----------</td>
</tr>
<tr>
<td>*</td>
<td>1 ST</td>
<td>W21X48</td>
<td>FAIL</td>
<td>Eq. H1-1b</td>
<td>1.439</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00</td>
<td>-79.80</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### SLENDERNESS

- Actual Slenderness Ratio: 57.946
- Allowable Slenderness Ratio: 300.000

#### STRENGTH CHECKS

- Critical L/C: 3
- Ratio: 1.439 (FAIL)
- Condition: Eq. H1-1b

#### DESIGN FORCES

- Fx: 0.000E+00
- Fy: 0.000E+00
- Fz: 1.200E+02
- Mx: 0.000E+00
- My: -7.980E+01
- Mz: 0.000E+00

#### SECTION PROPERTIES (UNIT: INCH)

- Azz: 7.000E+00
- Ayy: 7.210E+00
- Cw: 3.936E+03
- Szz: 9.311E+01
- Syy: 9.509E+00
- Izz: 9.590E+02
- Iyy: 3.870E+01
- Ix: 8.030E-01

#### MATERIAL PROPERTIES

- Fyld: 7199.999
- Fu: 9359.999

- Actual Member Length: 8.000

#### Design Parameters

- Kz: 1.00
- Ky: 1.00
- NSF: 1.00
- SLF: 1.00
- CSP: 12.00

#### SECTION CLASS

<table>
<thead>
<tr>
<th>UNSTIFFENED /</th>
<th>l</th>
<th>l p</th>
<th>l r</th>
<th>CASE</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>STIFFENED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression: Non-Slender</td>
</tr>
<tr>
<td>Slender</td>
</tr>
<tr>
<td>Flexure</td>
</tr>
<tr>
<td>Compact</td>
</tr>
</tbody>
</table>

---

**CHECK FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>6.35E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>3</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>6.87E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**Verification Examples**

**V.09 Steel Design**
### CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>$0.00E+00$</td>
<td>$5.54E+02$</td>
<td>$0.000$</td>
<td>Eq. E7-1</td>
<td>3</td>
</tr>
<tr>
<td>Min Buck</td>
<td>$0.00E+00$</td>
<td>$4.66E+02$</td>
<td>$0.000$</td>
<td>Eq. E7-1</td>
<td>3</td>
</tr>
<tr>
<td>Flexural</td>
<td>Tor Buck</td>
<td>$0.00E+00$</td>
<td>$4.97E+02$</td>
<td>$0.000$</td>
<td>Eq. E7-1</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Eff Area
- KL/r
- Fcr
- Fe
- Pn

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>$8.62E-02$</td>
<td>$11.64$</td>
<td>$6.28E+03$</td>
<td>$3.12E+05$</td>
<td>$6.15E+02$</td>
</tr>
<tr>
<td>Min Buck</td>
<td>$8.94E-02$</td>
<td>$57.95$</td>
<td>$5.28E+03$</td>
<td>$1.26E+04$</td>
<td>$5.17E+02$</td>
</tr>
<tr>
<td>Flexural</td>
<td>Tor Buck</td>
<td>$8.82E-02$</td>
<td>$5.64E+03$</td>
<td>$5.52E+02$</td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>$-1.20E+02$</td>
<td>$1.89E+02$</td>
<td>$0.635$</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
<tr>
<td>Local-Y</td>
<td>$0.00E+00$</td>
<td>$2.16E+02$</td>
<td>$0.000$</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate Results:

- Aw
- Cv
- Kv
- h/tw
- Vn

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>$4.86E-02$</td>
<td>$1.00$</td>
<td>$1.20$</td>
<td>$9.47$</td>
<td>$2.10E+02$</td>
</tr>
<tr>
<td>Local-Y</td>
<td>$5.01E-02$</td>
<td>$1.00$</td>
<td>$0.00$</td>
<td>$53.54$</td>
<td>$2.16E+02$</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>$0.00E+00$</td>
<td>$4.01E+02$</td>
<td>$0.000$</td>
<td>Eq. F2-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>$-7.98E+01$</td>
<td>$5.59E+01$</td>
<td>$1.428$</td>
<td>Eq. F6-1</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate Mn:

- My

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>$4.46E+02$</td>
<td>$0.00E+00$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>$6.21E+01$</td>
<td>$0.00E+00$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>$0.00E+00$</td>
<td>$3.71E+02$</td>
<td>$0.000$</td>
<td>Eq. F2-2</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>Me</td>
<td>$C_b$</td>
<td>$L_p$</td>
<td>$L_r$</td>
<td>$L_b$</td>
</tr>
<tr>
<td>Major</td>
<td>$4.12E+02$</td>
<td>$0.00E+00$</td>
<td>$1.00$</td>
<td>$5.93$</td>
<td>$16.76$</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-FLANGE LOCAL BUCKLING

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>$0.00E+00$</td>
<td>$3.99E+02$</td>
<td>$0.000$</td>
<td>Eq. F3-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>$-7.98E+01$</td>
<td>$5.55E+01$</td>
<td>$1.439$</td>
<td>Eq. F6-2</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate Fcr:

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>$4.44E+02$</td>
<td>$0.00E+00$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>$6.16E+01$</td>
<td>$0.00E+00$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th>Flexure Comp</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.439</td>
<td>Eq. H1-1b</td>
<td>3</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>1.439</td>
<td>Eq. H1-1b</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mcx /</td>
<td>Mrx /</td>
<td>Pc /</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Mcy</td>
<td>Mry</td>
<td>Pr</td>
<td></td>
</tr>
<tr>
<td>Flexure Comp</td>
<td>3.71E+02</td>
<td>0.00E+00</td>
<td>4.66E+02</td>
<td></td>
</tr>
<tr>
<td></td>
<td>5.55E+01</td>
<td>-7.98E+01</td>
<td>0.00E+00</td>
<td></td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>3.71E+02</td>
<td>0.00E+00</td>
<td>6.35E+02</td>
<td></td>
</tr>
<tr>
<td></td>
<td>5.55E+01</td>
<td>-7.98E+01</td>
<td>0.00E+00</td>
<td></td>
</tr>
</tbody>
</table>

**46. LOAD LIST 4**

**47. PARAMETER 2**

**48. CODE AISC UNIFIED 2010**

**49. METHOD ASD**

**50. FYLD 7200**

**51. FU 9360**

**52. TRACK 2 ALL**

**53. CHECK CODE ALL**

STEEL DESIGN

--- PAGE NO. ---

STAAD SPACE

--- PAGE NO. ---

STAAD.PRO CODE CHECKING - (AISC-360-10-ASD) v1.4a

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)

MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION

* 1 ST W21X48 (AISC SECTIONS)

   FAIL Eq. H1-1b 1.442 4
   0.00 -53.20 0.00 0.00

SLENDERNESS

Actual Slenderness Ratio : 57.946 L/C : 4
Allowable Slenderness Ratio : 300.000 LOC : 0.00

STRENGTH CHECKS

Critical L/C : 4 Ratio : 1.442(FAIL)
Loc : 0.00 Condition : Eq. H1-1b

DESIGN FORCES

Fx: 0.000E+00( ) Fy: 0.000E+00 Fz: 8.000E+01
Mx: 0.000E+00 My: -5.320E+01 Mz: 0.000E+00

SECTION PROPERTIES (UNIT: INCH)

Azz: 7.000E+00 Ayy: 7.210E+00 Cw: 3.936E+03
Szz: 9.311E+01 Syy: 9.509E+00
Izz: 9.590E+02 Iyy: 3.870E+01 Ix: 8.030E-01

MATERIAL PROPERTIES

Fyld: 7199.999 Fu: 9359.999

Actual Member Length: 8.000

Design Parameters

Kz: 1.00 Ky: 1.00 NSF: 1.00 SLF: 1.00 CSP: 12.00

SECTION CLASS UNSTIFFENED / l l p l r CASE

STIFFENED

Compression: Non-Slender 9.47 N/A 13.66 T.B4.1(a)-1
Slender 53.54 N/A 36.33 T.B4.1(a)-5

VERIFICATION EXAMPLES

STAAD.Pro 3928 User Manual
| Flexure : Non-Compact 9.47 9.27 24.39 T.B4.1(b)-10 |
|Compact 53.54 91.69 139.00 T.B4.1(b)-15 |

STAAD SPACE

STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD) v1.4a

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)

-- PAGE NO.

---

CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>4.22E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>4</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>4.58E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>3.68E+02</td>
<td>0.000</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>3.10E+02</td>
<td>0.000</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
<tr>
<td>Flexural</td>
<td>0.00E+00</td>
<td>3.31E+02</td>
<td>0.000</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
<tr>
<td>Tor Buck</td>
<td>0.00E+00</td>
<td>5.64E+03</td>
<td>5.52E+02</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
</tbody>
</table>

Results

<table>
<thead>
<tr>
<th>Eff Area</th>
<th>KL/r</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>8.62E-02</td>
<td>11.64</td>
<td>6.28E+03</td>
<td>3.12E+05</td>
</tr>
<tr>
<td>Min Buck</td>
<td>8.04E-02</td>
<td>57.95</td>
<td>5.28E+03</td>
<td>1.26E+04</td>
</tr>
</tbody>
</table>

CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>-8.00E+01</td>
<td>1.26E+02</td>
<td>0.636</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>1.44E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
</tbody>
</table>

Intermediate

<table>
<thead>
<tr>
<th>Aw</th>
<th>Cv</th>
<th>Kv</th>
<th>h/tw</th>
<th>Vn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>4.86E-02</td>
<td>1.00</td>
<td>1.20</td>
<td>9.47</td>
</tr>
<tr>
<td>Local-Y</td>
<td>5.01E-02</td>
<td>1.00</td>
<td>0.00</td>
<td>53.54</td>
</tr>
</tbody>
</table>

CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>2.67E+02</td>
<td>0.000</td>
<td>Eq. F2-1</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>-5.32E+01</td>
<td>3.72E+01</td>
<td>1.431</td>
<td>Eq. F6-1</td>
<td>4</td>
</tr>
</tbody>
</table>

Intermediate

<table>
<thead>
<tr>
<th>Mn</th>
<th>My</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>4.46E+02</td>
</tr>
<tr>
<td>Minor</td>
<td>6.21E+01</td>
</tr>
</tbody>
</table>

CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>2.47E+02</td>
<td>0.000</td>
<td>Eq. F2-2</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
</tr>
<tr>
<td>Major</td>
<td>4.12E+02</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>5.93</td>
<td>16.76</td>
</tr>
</tbody>
</table>

---

STAAD SPACE

STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD) v1.4a

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)
### V. AISC 360-10 - D-2

**Reference**


**Note:** This reference is freely available for download from the AISC website at [https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf](https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf)

**Problem**

From the reference:

Verify, by both ASD and LRFD, the tensile strength of an L4×4×1/2, ASTM A36 [...]. The member carries a dead load of 20 kips and a live load of 60 kips in tension. Calculate at what length this tension member would cease to satisfy the recommended slenderness limit. Assume that connection limit states do not govern.

Refer to the reference for calculations.

**Comparison**

*Table 545: Comparison of results*

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi_t P_n$ (kips) [LRFD]</td>
<td>122</td>
<td>122</td>
<td>none</td>
</tr>
<tr>
<td>$P_n / \Omega_t$ (kips) [ASD]</td>
<td>80.8</td>
<td>80.8</td>
<td>none</td>
</tr>
</tbody>
</table>
To calculate the length at which the member would exceed the slenderness limit of 300: (6 in)/(slenderness ratio) × 300

\[
(6 \text{ in})/(7.732) \times 300 = 232.8 \text{ in} = 19.4 \text{ ft}
\]

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - D-2.STD is typically installed with the program.

Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36.7236 FU 59.1464 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L40408
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
JOINT LOAD
2 FX 20
LOAD 2 LOADTYPE Live TITLE LOAD CASE 2
JOINT LOAD
2 FX 60
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
```
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

1

********************************************
*                                                *
*           STAAD.Pro CONNECT Edition           *
*           Version 22.01.00.**                 *
*           Proprietary Program of              *
*           Bentley Systems, Inc.               *
*           Date= APR 14, 2019                 *
*           Time= 23:11: 9                     *
*                                                *
* Licensed to: Bentley Systems Inc              *
********************************************

1. STAAD SPACE
INPUT FILE: AISC 360-10 - D-2.STD
2. START JOB INFORMATION
3. ENGINEER DATE 25-OCT-12
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT INCHES KIP
7. JOINT COORDINATES
8. 1 0 0 0; 2 6 0 0
9. MEMBER INCIDENCES
10. 1 1 2
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 29732.7
14. POISSON 0.3
15. DENSITY 0.000283
16. ALPHA 1.2E-05
17. DAMP 0.03
18. TYPE STEEL
19. STRENGTH FY 36.7236 FU 59.1464 RY 1.5 RT 1.2
20. END DEFINE MATERIAL
21. MEMBER PROPERTY AMERICAN
22. 1 TABLE ST L40408
23. CONSTANTS
24. MATERIAL STEEL ALL
25. SUPPORTS
26. 1 FIXED
27. LOAD 1 LOADTYPE DEAD TITLE LOAD CASE 1
28. JOINT LOAD
29. 2 FX 20
30. LOAD 2 LOADTYPE LIVE TITLE LOAD CASE 2
### Problem Statistics

<table>
<thead>
<tr>
<th>Category</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Joints</td>
<td>2</td>
</tr>
<tr>
<td>Number of Members</td>
<td>1</td>
</tr>
<tr>
<td>Number of Plates</td>
<td>0</td>
</tr>
<tr>
<td>Number of Solids</td>
<td>0</td>
</tr>
<tr>
<td>Number of Supports</td>
<td>1</td>
</tr>
</tbody>
</table>

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

Total primary load cases = 2, total degrees of freedom = 6
Total load combination cases = 2 so far.

### Steel Design

<table>
<thead>
<tr>
<th>Member</th>
<th>Table</th>
<th>Result/</th>
<th>Critical Cond/</th>
<th>Ratio/</th>
<th>Loading/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>Location</td>
<td></td>
<td></td>
</tr>
<tr>
<td>L40408</td>
<td>(AISC SECTIONS)</td>
<td>Pass</td>
<td>Eq. H2-1</td>
<td>0.988</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>120.00 T</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member Properties (Unit: Inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Azz: 1.333E+00</td>
</tr>
<tr>
<td>Szz: 3.121E+00</td>
</tr>
<tr>
<td>Izz: 2.258E+00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld: 36.000</td>
</tr>
<tr>
<td>Fu: 58.000</td>
</tr>
</tbody>
</table>

Actual Member Length: 6.000
### Design Parameters

<table>
<thead>
<tr>
<th>Kz:</th>
<th>1.00</th>
<th>Ky:</th>
<th>1.00</th>
<th>NSF:</th>
<th>1.00</th>
<th>SLF:</th>
<th>1.00</th>
<th>CSP:</th>
<th>12.00</th>
</tr>
</thead>
</table>

### SECTION CLASS

<table>
<thead>
<tr>
<th>UNSTIFFENED / STIFFENED</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
</tr>
</tbody>
</table>

#### Compression:
- Non-Slender: 8.00 N/A 12.93 T.B4.1(a)-3
- Slender: N/A N/A N/A N/A N/A

#### Flexure:
- Compact: 8.00 15.52 26.15 T.B4.1(b)-12
- Slender: N/A N/A N/A N/A N/A

### CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>1.20E+02</td>
<td>1.22E+02</td>
<td>0.988</td>
<td>Eq. D2-1</td>
<td>3</td>
</tr>
<tr>
<td>Rupture</td>
<td>1.20E+02</td>
<td>1.63E+02</td>
<td>0.736</td>
<td>Eq. D2-2</td>
<td>3</td>
</tr>
</tbody>
</table>

### CHECK FOR AXIAL COMPRESSION

**Min Buck**
- Eff Area: 3.75E+00
- KL/r: 63.94
- Fcr: 2.92E+01
- Fe: 7.18E+01
- Pn: 1.09E+02

### CHECK FOR SHEAR

**Local-Z**
- Aw: 1.75E+00
- Cv: 1.00
- Kv: 1.20
- h/tw: 8.00
- Vn: 3.78E+01

**Local-Y**
- Aw: 2.00E+00
- Cv: 1.00
- Kv: 1.20
- h/tw: 8.00
- Vn: 4.32E+01

### CHECK FOR BENDING-YIELDING

**Major**
- 0.00E+00
- 1.52E+02
- 0.000
- Eq. F10-1
- 3
- 0.00

**Minor**
- 0.00E+00
- 6.66E+01
- 0.000
- Eq. F10-1
- 3
- 0.00

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

**Major**
- 1.69E+02
- 1.12E+02
- Mn: 1.00
- Me: 1.00
- Cb: 0.00
- Lp: 0.00
- Lr: 0.00
- Lb: 6.00

---

**Verification Examples**

**V.09 Steel Design**

STAAD.Pro 3934

**User Manual**
### Verification Examples

**V.09 Steel Design**

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)**

#### CHECK FOR SHEAR AND NORMAL STRESS INTERACTION

<table>
<thead>
<tr>
<th>STRESS</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear</td>
<td>1.94E+01</td>
<td>0.000</td>
<td>Eq. H3-8</td>
<td>3</td>
</tr>
</tbody>
</table>

#### CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.000</td>
<td>Eq. H2-1</td>
<td>3</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.988</td>
<td>Eq. H2-1</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>frbw / frbz / fra / Fcbw Fcbz Fca</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flexure Comp</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>3.20E+01</td>
</tr>
</tbody>
</table>

#### GEOMETRIC AXIS DESIGN

#### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>9.60E+01</td>
<td>0.000</td>
<td>Eq. F10-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>9.60E+01</td>
<td>0.000</td>
<td>Eq. F10-1</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>My</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Major</td>
<td>1.07E+02</td>
<td>7.11E+01</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>1.07E+02</td>
<td>7.11E+01</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>7.68E+01</td>
<td>0.000</td>
<td>Eq. F10-3</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>7.68E+01</td>
<td>0.000</td>
<td>Eq. F10-3</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
</tr>
<tr>
<td>Major</td>
<td>8.53E+01</td>
<td>1.40E+05</td>
<td>1.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>Minor</td>
<td>8.53E+01</td>
<td>1.40E+05</td>
<td>1.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### STAAD SPACE -- PAGE NO. 6

44. LOAD LIST 4  
45. PARAMETER 2  
46. CODE AISC UNIFIED 2010  
47. METHOD ASD  
48. TRACK 2 ALL  
49. CHECK CODE ALL  

### STEEL DESIGN

### STAAD SPACE -- PAGE NO. 7

STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD) v1.4a

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST</td>
<td>L40408</td>
<td>PASS</td>
<td>Eq. H2-1</td>
<td>0.990</td>
<td>4</td>
<td>80.00</td>
</tr>
</tbody>
</table>
## SLENDERNESS
- Actual Slenderness Ratio: 7.732 L/C: 4
- Allowable Slenderness Ratio: 300.000 LOC: 0.00

## STRENGTH CHECKS
- Critical L/C: 4 Ratio: 0.990 (PASS)
  - Loc: 0.00 Condition: Eq. H2-1

## DESIGN FORCES
- Fx: 8.00E+01 (T) Fy: 0.000E+00 Fz: 0.000E+00
- Mx: 0.000E+00 My: 0.000E+00 Mz: 0.000E+00

## SECTION PROPERTIES (UNIT: INCH)
- Azz: 1.333E+00 Ayy: 1.333E+00 Cw: 3.662E-01
- Szz: 3.121E+00 Syy: 1.371E+00
- Izz: 2.258E+00 Iyy: 8.865E+00 Ix: 3.125E-01

## MATERIAL PROPERTIES
- Fyld: 36.000 Fu: 58.000

## Actual Member Length: 6.000

## Design Parameters
- Kz: 1.00 Ky: 1.00 NSF: 1.00 SLF: 1.00 CSP: 12.00

## SECTION CLASS UNSTIFFENED / STIFFENED

### STIFFENED
- Compression: Non-Slender 8.00 N/A 12.93 T.B4.1(a)-3
- Flexure: Compact 8.00 15.52 26.15 T.B4.1(b)-12

## STAAD SPACE
- STAAD.Pro Code Checking - (AISC-360-10-ASD) v1.4a

### ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)

## CHECK FOR AXIAL TENSION
- Yield 8.00E+01 8.08E+01 0.990 Eq. D2-1 4 0.00
- Rupture 8.00E+01 1.09E+02 0.736 Eq. D2-2 4 0.00

## CHECK FOR AXIAL COMPRESSION
- Min Buck 0.00E+00 6.55E+01 0.000 Eq. E3-1 4 0.00
- Intermediate

## CHECK FOR SHEAR
- Local-Z 0.00E+00 2.26E+01 0.000 Eq. G2-1 4 0.00
- Local-Y 0.00E+00 2.59E+01 0.000 Eq. G2-1 4 0.00
- Intermediate
### Results

<table>
<thead>
<tr>
<th></th>
<th>( Aw )</th>
<th>( Cv )</th>
<th>( Kv )</th>
<th>( h/tw )</th>
<th>( Vn )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>1.75E+00</td>
<td>1.00</td>
<td>1.20</td>
<td>8.00</td>
<td>3.78E+01</td>
</tr>
<tr>
<td>Local-Y</td>
<td>2.00E+00</td>
<td>1.00</td>
<td>1.20</td>
<td>8.00</td>
<td>4.32E+01</td>
</tr>
</tbody>
</table>

### Check for Bending-Yielding

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>1.01E+02</td>
<td>0.000</td>
<td>Eq. F10-1</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>4.43E+01</td>
<td>0.000</td>
<td>Eq. F10-1</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>My</td>
<td>Major 1.69E+02</td>
<td>1.12E+02</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>7.40E+01</td>
<td>4.94E+01</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Check for Bending-Lateral Torsional Buckling

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>1.01E+02</td>
<td>0.000</td>
<td>Eq. F10-3</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>Me</td>
<td>Major 1.69E+02</td>
<td>9.12E+03</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>7.40E+01</td>
<td>4.94E+01</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>Me</td>
<td>Major 1.69E+02</td>
<td>9.12E+03</td>
<td>1.00</td>
<td></td>
</tr>
</tbody>
</table>

### Check for Shear and Normal Stress Interaction

<table>
<thead>
<tr>
<th>Stress</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear</td>
<td>1.29E+01</td>
<td>0.000</td>
<td>Eq. H3-8</td>
<td>4</td>
</tr>
</tbody>
</table>

### Check for Flexure Tens/Comp Interaction

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.000</td>
<td>Eq. H2-1</td>
<td>4</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.990</td>
<td>Eq. H2-1</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>My</td>
<td>Major 1.07E+02</td>
<td>7.11E+01</td>
</tr>
<tr>
<td>Flexure Comp</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>3.23E+01</td>
<td>2.23E+01</td>
<td>1.75E+01</td>
</tr>
</tbody>
</table>

### Geometric Axis Design

### Check for Bending-Yielding

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>6.38E+01</td>
<td>0.000</td>
<td>Eq. F10-1</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>6.38E+01</td>
<td>0.000</td>
<td>Eq. F10-1</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>My</td>
<td>Major 1.07E+02</td>
<td>7.11E+01</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Check for Bending-Lateral Torsional Buckling

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>5.11E+01</td>
<td>0.000</td>
<td>Eq. F10-3</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>5.11E+01</td>
<td>0.000</td>
<td>Eq. F10-3</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>Me</td>
<td>Major 8.53E+01</td>
<td>1.40E+05</td>
<td>1.00</td>
<td></td>
</tr>
</tbody>
</table>

All units are - KIP INCH (UNLESS OTHERWISE Noted)
### V. AISC 360-10 - F.2-2

#### Reference


**Note:** This reference is freely available for download from the AISC website at [https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf](https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf)

#### Problem

Check an C15x33.9 channel of ASTM A36 steel which serves as a roof edge beam. The simple span is 25 ft. The nominal loads on the member are a dead load 0.23 kip/ft and a uniform live load of 0.69 kip/ft. The beam is braced at fifth points and at ends.  

Refer to Example F.2-2B in the reference for calculations.

#### Comparison

**Table 546: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi_b M_n$ (kips) [LRFD]</td>
<td>131</td>
<td>130</td>
<td>0.8%</td>
</tr>
<tr>
<td>$M_n/\Omega_b$ (kips) [ASD]</td>
<td>86.8</td>
<td>86.7</td>
<td>negligible</td>
</tr>
</tbody>
</table>

#### STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - F.STD` is typically installed with the program.
Tip: You can copy and paste this content directly into a .std file to run in STAAD.Pro.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 02-Nov-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 5 0; 2 25 5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.23
LOAD 2 LOADTYPE None TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.69
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
UNT 5 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD ASD
UNT 5 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
Verification Examples

V.09 Steel Design

STAAD Output

1

********************************************
*                                      *
*           STAAD.Pro CONNECT Edition   *
*           Version 22.01.00.**          *
*           Proprietary Program of      *
*           Bentley Systems, Inc.       *
*           Date= APR 14, 2019           *
*           Time= 23:11:31               *
*                                      *
* Licensed to: Bentley Systems Inc      *
********************************************

1. STAAD SPACE

INPUT FILE: AISC 360-10 - F.2-2.STD

2. START JOB INFORMATION

3. ENGINEER DATE 02-NOV-12

4. END JOB INFORMATION

5. INPUT WIDTH 79

6. UNIT FEET KIP

7. JOINT COORDINATES

8. 1 0 5 0; 2 25 5 0

9. MEMBER INCIDENCES

10. 1 1 2

11. DEFINE MATERIAL START

12. ISOTROPIC STEEL

13. E 4.176E+06

14. POISSON 0.3

15. DENSITY 0.489024

16. ALPHA 6E-06

17. DAMP 0.03

18. TYPE STEEL

19. STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2

20. END DEFINE MATERIAL

21. MEMBER PROPERTY AMERICAN

22. 1 TABLE ST C15X33

23. CONSTANTS

24. MATERIAL STEEL ALL

25. SUPPORTS

26. 1 PINNED

27. 2 FIXED BUT FX MZ

28. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1

29. MEMBER LOAD

30. 1 UNI GY -0.23

31. LOAD 2 LOADTYPE NONE TITLE LOAD CASE 2

32. MEMBER LOAD

33. 1 UNI GY -0.69

34. LOAD COMB 3 COMBINATION LOAD CASE 3

35. 1 1.2 2 1.6

36. LOAD COMB 4 COMBINATION LOAD CASE 4

37. 1 1.0 2 1.0

38. PERFORM ANALYSIS

STAAD SPACE

--- PAGE NO. ---

--- PAGE NO. ---

PROBLEM STATISTICS

--- PAGE NO. ---
**STAAD Pro Code Checking** - (AISC-360-10-LRFD) v1.4a

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE NOTED)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
</tbody>
</table>

---

**SLENDERNESS**

Actual Slenderness Ratio : 333.952
Allowable Slenderness Ratio : 300.000

---

**STRENGTH CHECKS**

Critical L/C : 3
Ratio : 1.113 (FAIL)
Loc : 0.00
Condition : SLENDERNESS

---

**DESIGN FORCES**

Fx: 0.000E+00   Fy: 1.725E+01   Fz: 0.000E+00
Mx: 0.000E+00   My: 0.000E+00   Mz: 0.000E+00

---

**SECTION PROPERTIES (UNIT: INCH)**

Azz: 4.420E+00   Ayy: 6.000E+00   Cw: 3.555E+02
Szz: 4.200E+01   Syy: 3.178E+00
Izz: 3.150E+02   Iyy: 8.070E+00   Ix: 9.595E-01

---

**MATERIAL PROPERTIES**

Fyld: 5183.999   Fu: 8351.999

---

**Actual Member Length:** 25.000

**Design Parameters**

Kz: 1.00   Ky: 1.00   NSF: 1.00   SLF: 1.00   CSP: 12.00

---

**SECTION CLASS**

<table>
<thead>
<tr>
<th>UNSTIFFENED /</th>
<th>CASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>l p l r</td>
<td></td>
</tr>
</tbody>
</table>

**STIFFENED**

**Compression**

| Non-Slender | 5.23 | N/A   | 15.89 | T.B4.1(a)-1 |
| Non-Slender | 34.25 | N/A   | 42.29 | T.B4.1(a)-5 |

**Flexure**

| Compact | 5.23 | 10.79 | 28.38 | T.B4.1(b)-10 |
| Compact | 34.25 | 106.72 | 161.78 | T.B4.1(b)-15 |
## STAAD SPACE

### STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a

*********************************************************

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)**

### CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>3.24E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>4.35E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>2.79E+02</td>
<td>0.000</td>
<td>Eq. E3-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>2.03E+01</td>
<td>0.000</td>
<td>Eq. E3-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Flexural</td>
<td>Tor Buck</td>
<td>0.00E+00</td>
<td>2.03E+02</td>
<td>0.000</td>
<td>Eq. E4-1</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Eff Area</td>
<td>KL/r</td>
<td>Fcr</td>
<td>Fe</td>
<td>Pn</td>
<td></td>
</tr>
<tr>
<td>Maj Buck</td>
<td>6.94E-02</td>
<td>53.45</td>
<td>4.46E+03</td>
<td>1.44E+04</td>
<td>3.10E+02</td>
<td></td>
</tr>
<tr>
<td>Min Buck</td>
<td>6.94E-02</td>
<td>333.95</td>
<td>3.24E+02</td>
<td>3.70E+02</td>
<td>2.25E+01</td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>8.59E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>1.72E+01</td>
<td>1.17E+02</td>
<td>0.148</td>
<td>Eq. G2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Aw</td>
<td>Cv</td>
<td>Kv</td>
<td>h/tw</td>
<td>Vn</td>
<td></td>
</tr>
<tr>
<td>Local-Z</td>
<td>3.07E-02</td>
<td>1.00</td>
<td>1.20</td>
<td>5.23</td>
<td>9.55E+01</td>
<td></td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.17E-02</td>
<td>1.00</td>
<td>5.00</td>
<td>34.25</td>
<td>1.30E+02</td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>1.08E+02</td>
<td>1.37E+02</td>
<td>0.786</td>
<td>Eq. F2-1</td>
<td>3</td>
<td>12.50</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>1.37E+01</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>My</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Major</td>
<td>1.52E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>1.53E+01</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>1.08E+02</td>
<td>1.30E+02</td>
<td>0.828</td>
<td>Eq. F2-2</td>
<td>3</td>
<td>12.50</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
<td>Lb</td>
</tr>
<tr>
<td>Major</td>
<td>1.45E+02</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>3.74</td>
<td>14.26</td>
<td>5.00</td>
</tr>
</tbody>
</table>

### CHECK FOR FLEXURE TENS/COMP INTERACTION
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/</th>
<th>RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
</tr>
<tr>
<td>1 ST</td>
<td>C15X33</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**SLENDERNESS**
Actual Slenderness Ratio : 333.952  L/C : 4  
Allowable Slenderness Ratio : 300.000  LOC : 0.00

**STRENGTH CHECKS**
Critical L/C : 4  Ratio : 1.113(FAIL)  
Loc : 0.00  Condition : SLENDERNESS

**DESIGN FORCES**
Fx: 0.000E+00( )  Fy: 1.150E+01  Fz: 0.000E+00  
Mx: 0.000E+00  My: 0.000E+00  Mz: 0.000E+00

**SECTION PROPERTIES** (UNIT: INCH)
Azz: 4.420E+00  Ayy: 6.000E+00  Cw: 3.555E+02  
Szz: 4.200E+01  Syy: 3.178E+00  Izz: 8.070E+00  Iyy: 8.070E+00  Ixz: 9.595E-01

**MATERIAL PROPERTIES**
Fyld: 5183.999  Fu: 8351.999

Actual Member Length: 25.000  
Design Parameters
Kz: 1.00  Ky: 1.00  NSF: 1.00  SLF: 1.00  CSP: 12.00

**SECTION CLASS** UNSTIFFENED / STIFFENED  
Compressing: Non-Slender 5.23  N/A 15.89  T.B4.1(a)-1

---

**Verification Examples**
V.09 Steel Design

STAAD.Pro 3943 User Manual
<table>
<thead>
<tr>
<th>Non-Slender</th>
<th>34.25</th>
<th>N/A</th>
<th>42.29</th>
<th>T.B4.1(a)-5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure</td>
<td>Compact</td>
<td>5.23</td>
<td>10.79</td>
<td>28.38</td>
</tr>
<tr>
<td></td>
<td>Compact</td>
<td>34.25</td>
<td>106.72</td>
<td>161.78</td>
</tr>
</tbody>
</table>

STAAD SPACE

--- PAGE NO.

STAAD.PRO CODE CHECKING - (AISC-360-10-ASD) v1.4a

***********************************************************************

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)**

---

**CHECK FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>2.16E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>4</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>2.90E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

---

**CHECK FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>1.85E+02</td>
<td>0.000</td>
<td>Eq. E3-1</td>
<td>4</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>1.35E+01</td>
<td>0.000</td>
<td>Eq. E3-1</td>
<td>4</td>
</tr>
</tbody>
</table>

---

**RESULTS**

<table>
<thead>
<tr>
<th>Eff Area</th>
<th>KL/r</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>6.94E-02</td>
<td>53.45</td>
<td>4.46E+03</td>
<td>1.44E+04</td>
</tr>
<tr>
<td>Min Buck</td>
<td>6.94E-02</td>
<td>106.72</td>
<td>161.78</td>
<td>1.44E+04</td>
</tr>
</tbody>
</table>

---

**CHECK FOR SHEAR**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>5.72E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
<tr>
<td>Local-Y</td>
<td>1.15E+01</td>
<td>7.76E+01</td>
<td>0.148</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
</tbody>
</table>

---

**RESULTS**

<table>
<thead>
<tr>
<th>Aw</th>
<th>Cv</th>
<th>Kv</th>
<th>h/tw</th>
<th>Vn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>3.07E-02</td>
<td>1.00</td>
<td>1.20</td>
<td>5.23</td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.17E-02</td>
<td>1.00</td>
<td>5.00</td>
<td>34.25</td>
</tr>
</tbody>
</table>

---

**CHECK FOR BENDING-YIELDING**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>7.19E+01</td>
<td>9.13E+01</td>
<td>0.788</td>
<td>Eq. F2-1</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>9.14E+00</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>4</td>
</tr>
</tbody>
</table>

---

**CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>7.19E+01</td>
<td>8.67E+01</td>
<td>0.829</td>
<td>Eq. F2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

---

STAAD SPACE

--- PAGE NO.
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)

<table>
<thead>
<tr>
<th>CHECK FOR FLEXURE TENS/COMP INTERACTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>RAT</td>
</tr>
<tr>
<td>----------------</td>
</tr>
<tr>
<td>Flexure Comp</td>
</tr>
<tr>
<td>Flexure Tens</td>
</tr>
<tr>
<td>Intermediate</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Flexure Comp</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Flexure Tens</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

53. FINISH

*********** END OF THE STAAD.Pro RUN ***********

**** DATE= APR 14,2019    TIME= 23:11:32 ****

V. AISC 360-10 - H.1B

Reference


Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Objective

From the reference:

Using AISC Manual tables to determine the available compressive and flexural strengths, determine if an ASTM A992 W14×99 has sufficient available strength to support the axial forces and moments listed as follows, obtained from a second-order analysis that includes P-δ effects. The unbraced length is 14 ft and the member has pinned ends. KLx = KLy = Lb = 14.0 ft.

Refer to the reference for calculations.

Note: A static analysis is used in STAAD.Pro as the reference supplies only the resulting forces from an assumed P-δ analysis.
### Table 547: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>LRFD</strong></td>
<td>Interaction Ratio</td>
<td>0.928</td>
<td>0.929</td>
</tr>
<tr>
<td>$\phi M_{nx}$ (kip-ft)</td>
<td>642</td>
<td>649</td>
<td>1.1%</td>
</tr>
<tr>
<td>$\phi M_{ny}$ (kip-ft)</td>
<td>311</td>
<td>314</td>
<td>1.0%</td>
</tr>
<tr>
<td>$\phi \text{P}_n$ (kips)</td>
<td>1,130</td>
<td>1,130</td>
<td>none</td>
</tr>
<tr>
<td><strong>ASD</strong></td>
<td>Interaction Ratio</td>
<td>0.931</td>
<td>0.932</td>
</tr>
<tr>
<td>$M_{nx}/\Omega$ (kip-ft)</td>
<td>428</td>
<td>432</td>
<td>0.9%</td>
</tr>
<tr>
<td>$M_{ny}/\Omega$ (kip-ft)</td>
<td>207</td>
<td>209</td>
<td>1.0%</td>
</tr>
<tr>
<td>Compression (kips)</td>
<td>750</td>
<td>750</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - H.STD is typically installed with the program.

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Nov-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 14 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X99
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
1 FY -400 MX 80 MZ 250
LOAD 2 LOADTYPE None TITLE LOAD CASE 2
JOINT LOAD
1 FY -267 MX 53.3 MZ 167
PERFORM ANALYSIS
LOAD LIST 1
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
LZ 14 ALL
LY 14 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 2
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
LZ 14 ALL
LY 14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

1

******************************************************************************
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.* *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 23:11:41 *
* Licensed to: Bentley Systems Inc *
******************************************************************************
1. STAAD SPACE
INPUT FILE: AISC 360-10 - H.1B.STD
2. START JOB INFORMATION
3. ENGINEER DATE 19-NOV-12
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT FEET KIP
7. JOINT COORDINATES
8. 1 0 14 0; 2 0 0 0
9. MEMBER INCIDENCES
10. 1 1 2
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 4.176E+06
14. POISSON 0.3
15. DENSITY 0.489024
16. ALPHA 6E-06
17. DAMP 0.03
18. TYPE STEEL
19. STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
20. END DEFINE MATERIAL
21. MEMBER PROPERTY AMERICAN
22. 1 TABLE ST W14X99
23. CONSTANTS
24. MATERIAL STEEL ALL
25. SUPPORTS
26. 2 FIXED
27. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
28. JOINT LOAD
29. 1 FY -400 MX 80 MZ 250
30. LOAD 2 LOADTYPE NONE TITLE LOAD CASE 2
31. JOINT LOAD
32. 1 FY -267 MX 53.3 MZ 167
33. PERFORM ANALYSIS
34. LOAD LIST 1
35. PARAMETER 1
36. CODE AISC UNIFIED 2010
37. METHOD LRFD
38. FYLD 7200 ALL
39. FU 9360 ALL
40. LZ 14 ALL
41. LY 14 ALL
42. TRACK 2 ALL
43. CHECK CODE ALL

### PROBLEM STATISTICS

<table>
<thead>
<tr>
<th>NUMBER OF JOINTS</th>
<th>NUMBER OF MEMBERS</th>
<th>NUMBER OF PLATES</th>
<th>NUMBER OF SOLIDS</th>
<th>NUMBER OF SURFACES</th>
<th>SUPPORTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 6
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX MY MZ LOCATION</td>
</tr>
<tr>
<td>1 ST</td>
<td>W14X99</td>
<td>PAss Eq. H1-1a 0.929 1 400.00 C 80.00 250.00 0.00</td>
</tr>
</tbody>
</table>

### STEEL DESIGN

<table>
<thead>
<tr>
<th>SLENDERNESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actual Slenderness Ratio : 45.200 L/C : 1</td>
</tr>
<tr>
<td>Allowable Slenderness Ratio :</td>
</tr>
<tr>
<td>-------------------------------</td>
</tr>
</tbody>
</table>

### STRENGTH CHECKS

<table>
<thead>
<tr>
<th>Critical L/C :</th>
<th>1</th>
<th>Ratio :</th>
<th>0.929 (PASS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Loc :</td>
<td>0.00</td>
<td>Condition :</td>
<td>Eq. H1-1a</td>
</tr>
</tbody>
</table>

### DESIGN FORCES

<table>
<thead>
<tr>
<th>Fx :</th>
<th>4.000E+02 (C)</th>
<th>Fy :</th>
<th>0.000E+00</th>
<th>Fz :</th>
<th>0.000E+00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mx :</td>
<td>0.000E+00</td>
<td>My :</td>
<td>8.000E+01</td>
<td>Mz :</td>
<td>2.500E+02</td>
</tr>
</tbody>
</table>

### SECTION PROPERTIES (UNIT: INCH)

<table>
<thead>
<tr>
<th>Azz :</th>
<th>2.278E+01</th>
<th>Ayy :</th>
<th>6.887E+00</th>
<th>Cw :</th>
<th>1.810E+04</th>
</tr>
</thead>
<tbody>
<tr>
<td>Szz :</td>
<td>1.563E+02</td>
<td>Syy :</td>
<td>5.507E+01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Izz :</td>
<td>1.110E+03</td>
<td>Iyy :</td>
<td>4.020E+02</td>
<td>Ix :</td>
<td>5.370E+00</td>
</tr>
</tbody>
</table>

### MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Fyld :</th>
<th>7199.999</th>
<th>Fu :</th>
<th>9359.999</th>
</tr>
</thead>
</table>

### Actual Member Length: 14.000

### Design Parameters

<table>
<thead>
<tr>
<th>Kz :</th>
<th>1.00</th>
<th>Ky :</th>
<th>1.00</th>
<th>NSF :</th>
<th>1.00</th>
<th>SLF :</th>
<th>1.00</th>
<th>CSP :</th>
<th>12.00</th>
</tr>
</thead>
</table>

### SECTION CLASS

<table>
<thead>
<tr>
<th>STIFFENED /</th>
<th>UNSTIFFENED</th>
<th>l</th>
<th>l p</th>
<th>l r</th>
<th>CASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compression :</td>
<td>Non-Slender</td>
<td>9.36</td>
<td>N/A</td>
<td>13.49</td>
<td>T.B4.1(a)-1</td>
</tr>
<tr>
<td>Flexure :</td>
<td>Non-Compact</td>
<td>9.15</td>
<td>24.08</td>
<td>T.B4.1(b)-10</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Compact</td>
<td>90.55</td>
<td>137.27</td>
<td>T.B4.1(b)-15</td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>1.31E+03</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>1</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>1.42E+03</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>4.00E+02</td>
<td>1.24E+03</td>
<td>0.322</td>
<td>Eq. E3-1</td>
<td>1</td>
</tr>
<tr>
<td>Min Buck</td>
<td>4.00E+02</td>
<td>1.13E+03</td>
<td>0.355</td>
<td>Eq. E3-1</td>
<td>1</td>
</tr>
<tr>
<td>Flexural Tor Buck</td>
<td>4.00E+02</td>
<td>1.15E+03</td>
<td>0.348</td>
<td>Eq. E4-1</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>2.02E-01</td>
<td>27.20</td>
<td>6.82E+03</td>
<td>5.57E+04</td>
<td>1.38E+03</td>
</tr>
<tr>
<td>Min Buck</td>
<td>2.02E-01</td>
<td>45.20</td>
<td>6.20E+03</td>
<td>2.02E+04</td>
<td>1.25E+03</td>
</tr>
<tr>
<td>Flexural</td>
<td>Ag</td>
<td>Fcr</td>
<td>Pn</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tor Buck</td>
<td>2.02E-01</td>
<td>6.32E+03</td>
<td>1.28E+03</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### STAAD SPACE -- PAGE NO. 4

**STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a**

---

**Ver: 09 Steel Design**

---

**STAAD.Pro 3949 User Manual**
<table>
<thead>
<tr>
<th></th>
<th>0.00E+00</th>
<th>6.15E+02</th>
<th>0.000</th>
<th>Eq. G2-1</th>
<th>1</th>
<th>0.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>2.07E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Intermediate Results**

<table>
<thead>
<tr>
<th></th>
<th>A_w</th>
<th>C_v</th>
<th>K_v</th>
<th>h/t_w</th>
<th>V_n</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>1.58E-01</td>
<td>1.00</td>
<td>1.20</td>
<td>9.36</td>
<td>6.83E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.78E-02</td>
<td>1.00</td>
<td>0.00</td>
<td>23.59</td>
<td>2.07E+02</td>
</tr>
</tbody>
</table>

**CHECK FOR BENDING-YIELDING**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-2.50E+02</td>
<td>6.49E+02</td>
<td>0.385</td>
<td>Eq. F2-1</td>
<td>1</td>
</tr>
<tr>
<td>Minor</td>
<td>8.00E+01</td>
<td>3.14E+02</td>
<td>0.255</td>
<td>Eq. F6-1</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-2.50E+02</td>
<td>6.42E+02</td>
<td>0.389</td>
<td>Eq. F2-2</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECK FOR BENDING-FLANGE LOCAL BUCKLING**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-2.50E+02</td>
<td>6.45E+02</td>
<td>0.387</td>
<td>Eq. F3-1</td>
<td>1</td>
</tr>
<tr>
<td>Minor</td>
<td>8.00E+01</td>
<td>3.11E+02</td>
<td>0.257</td>
<td>Eq. F6-2</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECK FOR FLEXURE TENS/COMP INTERACTION**

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.929</td>
<td>Eq. H1-1a</td>
<td>1</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.646</td>
<td>Eq. H1-1b</td>
<td>1</td>
</tr>
</tbody>
</table>

**Verification Examples**

V.09 Steel Design

STAAD Pro

User Manual
### STEEL DESIGN

**STAAD SPACE**

--- PAGE NO.

6

**STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD) v1.4a**

---------------

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>1 ST</th>
<th>W14X99</th>
<th>PASS</th>
<th>Eq. H1-1a</th>
<th>0.932</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>267.00</td>
<td>C</td>
<td>53.30</td>
<td>167.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SLENDERNESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actual Slenderness Ratio : 45.200</td>
</tr>
<tr>
<td>Allowable Slenderness Ratio : 200.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>STRENGTH CHECKS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical L/C : 2</td>
</tr>
<tr>
<td>Loc : 0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DESIGN FORCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fx: 2.670E+02(C )</td>
</tr>
<tr>
<td>Mx: 0.000E+00</td>
</tr>
<tr>
<td>Mz: 1.670E+02</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SECTION PROPERTIES (UNIT: INCH)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Azz: 2.278E+01</td>
</tr>
<tr>
<td>Szz: 1.563E+02</td>
</tr>
<tr>
<td>Izz: 1.110E+03</td>
</tr>
<tr>
<td>Ix: 5.370E+00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>MATERIAL PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld: 7199.999</td>
</tr>
</tbody>
</table>

| Actual Member Length: 14.000 |
| Design Parameters |
| Kz: 1.00 | Ky: 1.00 |
| NSF: 1.00 | SLF: 1.00 | CSP: 12.00 |

<table>
<thead>
<tr>
<th>SECTION CLASS UNSTIFFENED /</th>
<th>l</th>
<th>l</th>
<th>p</th>
<th>1</th>
<th>r</th>
<th>CASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>STIFFENED</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Compression : Non-Slender : 9.36</td>
<td>N/A</td>
<td>13.49</td>
<td>T.B4.1(a)-1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Non-Slender : 23.59</td>
<td>N/A</td>
<td>35.88</td>
<td>T.B4.1(a)-5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flexure : Non-Compact : 9.36</td>
<td>9.15</td>
<td>24.08</td>
<td>T.B4.1(b)-10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Compact : 23.59</td>
<td>90.55</td>
<td>137.27</td>
<td>T.B4.1(b)-15</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

--- PAGE NO.

7

**STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD) v1.4a**

---------------

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>CHECK FOR AXIAL TENSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>FORCE</td>
</tr>
<tr>
<td>Yield</td>
</tr>
<tr>
<td>Rupture</td>
</tr>
</tbody>
</table>

| CHECK FOR AXIAL COMPRESSION |

---
## Force Capacity Ratio Criteria L/C LOC

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>2.67E+02</td>
<td>8.25E+02</td>
<td>Eq. E3-1</td>
<td>2</td>
<td>0.00</td>
</tr>
<tr>
<td>Min Buck</td>
<td>2.67E+02</td>
<td>7.50E+02</td>
<td>Eq. E3-1</td>
<td>2</td>
<td>0.00</td>
</tr>
<tr>
<td>Flexural Tor Buck</td>
<td>2.67E+02</td>
<td>7.65E+02</td>
<td>Eq. E4-1</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Intermediate Results

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>2.02E-01</td>
<td>27.20</td>
<td>6.82E+03</td>
<td>5.57E+04</td>
<td>1.38E+03</td>
</tr>
<tr>
<td>Min Buck</td>
<td>2.02E-01</td>
<td>45.20</td>
<td>6.20E+03</td>
<td>2.02E+04</td>
<td>1.25E+03</td>
</tr>
</tbody>
</table>

### Flexural Results

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>2.02E-01</td>
<td>6.32E+03</td>
<td>1.28E+03</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Checks for Shear

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>4.09E+02</td>
<td>Eq. G2-1</td>
<td>2</td>
<td>0.00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>1.38E+02</td>
<td>Eq. G2-1</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Intermediate Results

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>1.58E-01</td>
<td>1.00</td>
<td>1.20</td>
<td>9.36</td>
<td>6.83E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.78E-02</td>
<td>1.00</td>
<td>0.00</td>
<td>23.59</td>
<td>2.07E+02</td>
</tr>
</tbody>
</table>

### Checks for Bending-Yielding

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-1.67E+02</td>
<td>4.32E+02</td>
<td>Eq. F2-1</td>
<td>2</td>
<td>0.00</td>
</tr>
<tr>
<td>Minor</td>
<td>5.33E+01</td>
<td>2.09E+02</td>
<td>Eq. F6-1</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Intermediate Results

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>7.21E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>3.48E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Checks for Bending-Lateral Torsional Buckling

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-1.67E+02</td>
<td>4.27E+02</td>
<td>Eq. F2-2</td>
<td>2</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>7.14E+02</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>13.13</td>
<td>45.47</td>
</tr>
</tbody>
</table>

### Checks for Bending-Flange Local Buckling

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>-1.67E+02</td>
<td>4.29E+02</td>
<td>Eq. F3-1</td>
<td>2</td>
<td>0.00</td>
</tr>
<tr>
<td>Minor</td>
<td>5.33E+01</td>
<td>2.07E+02</td>
<td>Eq. F6-2</td>
<td>2</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Intermediate Results

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>7.17E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>3.46E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Flexure Tens/Comp Interaction

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.932</td>
<td>Eq. H1-1a</td>
<td>2</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.648</td>
<td>Eq. H1-1b</td>
<td>2</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

### Intermediate Results

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mcx /</td>
<td>Mcy /</td>
<td>Mrx /</td>
<td>Pc /</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mry /</td>
<td>Pr</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### V. AISC 360-10 - E.5

**Reference**


**Note:** This reference is freely available for download from the AISC website at [https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf](https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf)

**Problem**

From the reference:

Verify the strength of a 2L 4×3-1/2×3/8 LLBB (3/8-in. separation) strut, ASTM A36, with a length of 8 ft and pinned ends carrying an axial dead load of 20 kips and live load of 60 kips.

Refer to the reference for calculations.

**Notes**

STAAD.Pro does *not* perform design per Section E6 of the specification. It is assumed that sufficient intermediate connectors of an appropriate type are present. It is left to the engineer to verify the intermediate connector design.

**Comparison**

**Table 548: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\phi_c P_n) [LRFD]</td>
<td>128</td>
<td>127</td>
<td>0.8%</td>
</tr>
</tbody>
</table>
### Verification Examples

V.09 Steel Design

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_n/Ω_c$ (kips) [ASD]</td>
<td>85.0</td>
<td>84.6</td>
<td>0.5%</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - E.STD is typically installed with the program.

**Tip:** You can copy and paste this content directly into a .std file to run in STAAD.Pro.

```plaintext
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 5-Dec-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE LD L40356 SP 0.0625
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
1 FY -20
LOAD 2 LOADTYPE None TITLE LOAD CASE 2
JOINT LOAD
1 FY -60
LOAD COMB 3 ULTIMATE LOADS
1 1.2 2 1.6
LOAD COMB 4 SERVICE LOADS
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
```
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
LX 56 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 36 ALL
FU 58 ALL
LX 56 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

1. STAAD SPACE
INPUT FILE: AISC 360-10 - E.5.STD
2. START JOB INFORMATION
3. ENGINEER DATE 5-DEC-12
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT FEET KIP
7. JOINT COORDINATES
8. 1 0 8 0; 2 0 0 0
9. MEMBER INCIDENCES
10. 1 1 2
11. MEMBER PROPERTY AMERICAN
12. 1 TABLE LD L40356 SP 0.0625
13. UNIT INCHES KIP
14. DEFINE MATERIAL START
15. ISOTROPIC MATERIAL1
16. E 29000
17. POISSON 0.3
18. ISOTROPIC STEEL
19. E 29000
20. POISSON 0.3
21. DENSITY 0.000283
22. ALPHA 6E-06
23. DAMP 0.03
24. TYPE STEEL
25. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
26. END DEFINE MATERIAL
27. UNIT FEET KIP
28. CONSTANTS
29. MATERIAL STEEL ALL
30. SUPPORTS
31. 2 FIXED
32. LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
33. JOINT LOAD
34. 1 FY -20
35. LOAD 2 LOADTYPE NONE TITLE LOAD CASE 2
36. JOINT LOAD
37. 1 FY -60
38. LOAD COMB 3 ULTIMATE LOADS
39. UNIT FEET KIP
40. LOAD COMB 4 SERVICE LOADS
41. 1 1.0 2 1.0
42. PERFORM ANALYSIS
43. PROBLEM STATISTICS
\begin{itemize}
\item NUMBER OF JOINTS: 2
\item NUMBER OF MEMBERS: 1
\item NUMBER OF PLATES: 0
\item NUMBER OF SOLIDS: 0
\item NUMBER OF SURFACES: 0
\item NUMBER OF SUPPORTS: 1
\end{itemize}
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 6
TOTAL LOAD COMBINATION CASES = 2

44. UNIT INCHES KIP
45. LOAD LIST 3
46. PARAMETER 1
47. CODE AISC UNIFIED 2010
48. METHOD LRFD
49. FYLD 36 ALL
50. FU 58 ALL
51. LX 56 ALL
52. TRACK 2 ALL
53. CHECK CODE ALL

STEEL DESIGN

49. STAAD SPACE

50. STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a
*****************************************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
     #   #   FX   MY   MZ   LOCATION
=======================================================================
1  LD L40356 (AISC SECTIONS) PASS Eq. H2-1 0.943 3
  120.00 C 0.00 0.00 0.00
\begin{itemize}
\item SLENDERNESS
\item Actual Slenderness Ratio : 76.914 L/C : 3
\item Allowable Slenderness Ratio : 200.000 LOC : 0.00
\end{itemize}
# STRENGTH CHECKS

**Critical L/C :** 3  **Ratio :** 0.943 (PASS)

Loc : 0.00  **Condition :** Eq. H2-1

---

## DESIGN FORCES

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

## SELECTION PROPERTIES (UNIT: INCH)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Azz</td>
<td>2.625E+00</td>
</tr>
<tr>
<td>Ayy</td>
<td>1.500E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>Szz</td>
<td>6.950E+00</td>
</tr>
<tr>
<td>Syy</td>
<td>3.978E+00</td>
</tr>
<tr>
<td>Izz</td>
<td>8.350E+00</td>
</tr>
<tr>
<td>Iyy</td>
<td>1.541E+01</td>
</tr>
<tr>
<td>Ix</td>
<td>2.512E-01</td>
</tr>
</tbody>
</table>

---

## MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld</td>
<td>36.000</td>
</tr>
<tr>
<td>Fu</td>
<td>58.000</td>
</tr>
</tbody>
</table>

---

## Actual Member Length:

96.000

---

## Design Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kz</td>
<td>1.00</td>
</tr>
<tr>
<td>Ky</td>
<td>1.00</td>
</tr>
<tr>
<td>NSF</td>
<td>1.00</td>
</tr>
<tr>
<td>SLF</td>
<td>1.00</td>
</tr>
<tr>
<td>CSP</td>
<td>12.00</td>
</tr>
</tbody>
</table>

---

## SECTION CLASS

STIFFENED / UNSTIFFENED

- **Compression:** Non-Slender
  - 10.67
  - N/A
  - 12.77
  - T.B4.1(a)-3
- **Flexure:** Compact
  - 9.33
  - 15.33
  - 25.83
  - T.B4.1(b)-12

---

## RESULTS

<table>
<thead>
<tr>
<th>Category</th>
<th>Eff Area</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>5.36E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Min Buck</td>
<td>5.36E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flexural</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tor Buck</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

## CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>Category</th>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>5.10E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>2.92E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)

---

## STAAD SPACE

STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)  v1.4a

********************************************
### Intermediate Results

<table>
<thead>
<tr>
<th></th>
<th>$A_w$</th>
<th>$C_v$</th>
<th>$K_v$</th>
<th>$h/t_w$</th>
<th>$V_m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>2.62E+00</td>
<td>1.00</td>
<td>1.20</td>
<td>9.33</td>
<td>5.67E+01</td>
</tr>
<tr>
<td>Local-Y</td>
<td>1.50E+00</td>
<td>1.00</td>
<td>1.20</td>
<td>10.67</td>
<td>3.24E+01</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>1.55E+02</td>
<td>0.000</td>
<td>Eq. F9-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>2.06E+02</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>My</td>
<td>Major</td>
<td>1.72E+02</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Minor</td>
<td>2.29E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>2.09E+03</td>
<td>0.000</td>
<td>Eq. F9-4</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
</tr>
<tr>
<td>Major</td>
<td>2.32E+03</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECK FOR FLEXURE TENS/CMP INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.943</td>
<td>Eq. H2-1</td>
<td>3</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.000</td>
<td>Eq. H2-1</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate</td>
<td>frbw /</td>
<td>frbz /</td>
<td>fra /</td>
</tr>
<tr>
<td>Fcbw</td>
<td>Fcbz</td>
<td>Fca</td>
<td></td>
</tr>
<tr>
<td>Flexure Comp</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>2.24E+01</td>
</tr>
<tr>
<td>5.18E+01</td>
<td>5.18E+01</td>
<td>2.37E+01</td>
<td></td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>2.23E+01</td>
<td>5.18E+01</td>
<td>3.24E+01</td>
<td></td>
</tr>
</tbody>
</table>
**SLENDERNESS**

- Actual Slenderness Ratio: 76.914  L/C: 4
- Allowable Slenderness Ratio: 200.000  LOC: 0.00

**STRENGTH CHECKS**

- Critical L/C: 4  Ratio: 0.945 (PASS)
- Loc: 0.00  Condition: Eq. H2-1

**DESIGN FORCES**

- Fx: 8.000E+01(C)  Fy: 0.000E+00  Fz: 0.000E+00
- Mx: 0.000E+00  My: 0.000E+00  Mz: 0.000E+00

**SECTION PROPERTIES (UNIT: INCH)**

- Azz: 2.625E+00  Ayy: 1.500E+00  Cw: 0.000E+00
- Szz: 6.950E+00  Syy: 3.978E+00
- Izz: 8.350E+00  Iyy: 1.541E+01  Ix: 2.512E-01

**MATERIAL PROPERTIES**

- Fyld: 36.000  Fu: 58.000

**Actual Member Length:** 96.000

**Design Parameters**

- Kz: 1.00  Ky: 1.00  NSF: 1.00  SLF: 1.00  CSP: 12.00

**SECTION CLASS UNSTIFFENED / STIFFENED**

<table>
<thead>
<tr>
<th>Compression</th>
<th>Flexure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Non-Slender</td>
<td>Compact</td>
</tr>
<tr>
<td>10.67</td>
<td>9.33</td>
</tr>
<tr>
<td>N/A</td>
<td>15.33</td>
</tr>
<tr>
<td>12.77</td>
<td>25.83</td>
</tr>
<tr>
<td>T.B4.1(a)-3</td>
<td>T.B4.1(b)-12</td>
</tr>
</tbody>
</table>

**CHECK FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>1.16E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>4</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>1.55E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

**CHECK FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>8.00E+01</td>
<td>8.46E+01</td>
<td>0.945</td>
<td>Eq. E3-1</td>
<td>4</td>
</tr>
<tr>
<td>Min Buck</td>
<td>8.00E+01</td>
<td>9.76E+01</td>
<td>0.820</td>
<td>Eq. E3-1</td>
<td>4</td>
</tr>
<tr>
<td>Flexural</td>
<td>8.00E+01</td>
<td>9.05E+01</td>
<td>0.884</td>
<td>Eq. E4-1</td>
<td>4</td>
</tr>
<tr>
<td>Tor Buck</td>
<td>0.00E+00</td>
<td>2.82E+01</td>
<td>1.51E+02</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Results</td>
<td>Eff Area</td>
<td>KL/r</td>
<td>Fcr</td>
<td>Fe</td>
<td>Pn</td>
</tr>
<tr>
<td>Maj Buck</td>
<td>5.36E+00</td>
<td>76.91</td>
<td>2.64E+01</td>
<td>4.84E+01</td>
<td>1.41E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>5.36E+00</td>
<td>56.61</td>
<td>3.04E+01</td>
<td>8.93E+01</td>
<td>1.63E+02</td>
</tr>
<tr>
<td>Flexural</td>
<td>Ag</td>
<td>Fcr</td>
<td>Pn</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tor Buck</td>
<td>0.00E+00</td>
<td>2.82E+01</td>
<td>1.51E+02</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>3.40E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>1.94E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Results</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Local-Z</td>
<td>2.62E+00</td>
<td>1.00</td>
<td>1.20</td>
<td>9.33</td>
<td>5.67E+01</td>
<td></td>
</tr>
<tr>
<td>Local-Y</td>
<td>1.50E+00</td>
<td>1.00</td>
<td>1.20</td>
<td>10.67</td>
<td>3.24E+01</td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>1.03E+02</td>
<td>0.000</td>
<td>Eq. F9-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>1.37E+02</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>My</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Major</td>
<td>1.72E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td>96.00</td>
<td></td>
</tr>
<tr>
<td>Minor</td>
<td>2.29E+02</td>
<td>0.00E+00</td>
<td></td>
<td></td>
<td>96.00</td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th></th>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>1.39E+03</td>
<td>0.000</td>
<td>Eq. F9-4</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>Mn</td>
<td>Me</td>
<td>Cb</td>
<td>Lp</td>
<td>Lr</td>
<td>Lb</td>
</tr>
<tr>
<td>Major</td>
<td>2.32E+03</td>
<td>0.00E+00</td>
<td>1.00</td>
<td>0.00</td>
<td>0.00</td>
<td>96.00</td>
</tr>
</tbody>
</table>

### CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th></th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.945</td>
<td>Eq. H2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.000</td>
<td>Eq. H2-1</td>
<td>4</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate</td>
<td>frbw / Frbz / Fra /</td>
<td>Fcbw</td>
<td>Fcbz</td>
<td>Fca</td>
</tr>
<tr>
<td>Flexure Comp</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>1.49E+01</td>
<td></td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>1.48E+01</td>
</tr>
</tbody>
</table>

62. FINISH

********** END OF THE STAAD.Pro RUN **********

**** DATE= APR 14,2019 TIME= 23:11:20 ****
**Verification Examples**

V.09 Steel Design

---

### V. AISC 360-10 - F.1-3B

**Reference**


**Note:** This reference is freely available for download from the AISC website at [https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf](https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf)

**Problem**

Verify the available flexural strength of a W18x50, ASTM A992 beam. The span is 35 ft and the beam is braced at both ends and at the center. The dead load is 0.45 kip/ft and the live load is 0.75 kip/ft.

**Comparison**

**Table 549: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi_b M_n$ (ft·kips) [LRFD]</td>
<td>289</td>
<td>3,460 in-kips = 288.3</td>
<td>negligible</td>
</tr>
<tr>
<td>$M_n/O_b$ (ft·kips) [ASD]</td>
<td>191</td>
<td>2,300 in-kips = 191.7</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - F.STD` is typically installed with the program.

**Tip:** You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

---

**STAAD SPACE**

START JOB INFORMATION
ENGINEER DATE 25-OCT-12
JOB NAME AISC 360-10 Validation
JOB CLIENT Bentley
JOB NO 7
END JOB INFORMATION

******************************************************************************
* AISC Example F.1-3
* W-SHAPE FLEXURAL MEMBER DESIGN IN STRONG-AXIS BENDING,
* BRACED AT MIDSPAN
* (1 analytical member with 2 unbraced segment)
******************************************************************************

INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
  1 0 0 0; 2 420 0 0;
MEMBER INCIDENCES
  1 1 2;

******************************************************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 59 RY 1.5 RT 1.2
END DEFINE MATERIAL

********************
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
********************

CONSTANTS
MATERIAL STEEL ALL
********************

SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
********************

LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.0375
**
LOAD 2 LOADTYPE Live TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.0625
**
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
**
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
********************

PERFORM ANALYSIS
**

LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
*
* ASTM A992
FYLD 50 ALL
FU 65 ALL
*CB 0 ALL
CB 1.3 ALL
*
LX 210 ALL
UNT 210 ALL
UNB 210 ALL
*
MAIN 1 ALL
*
TRACK 2 ALL
CHECK CODE ALL
********************
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
*  
* ASTM A992
FYLD 50 ALL
FU 65 ALL
*CB 0 ALL
CB 1.3 ALL
*  
LX 210 ALL
UNT 210 ALL
UNB 210 ALL
*  
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

1

********************************************************************************
*           STAAD.Pro CONNECT Edition              *
*           Version 22.01.00.**                    *
*           Proprietary Program of                 *
*           Bentley Systems, Inc.                  *
*           Date= APR 14, 2019                    *
*           Time= 23:11:24                        *
*                                                  *
*  Licensed to: Bentley Systems Inc               *
********************************************************************************
1. STAAD SPACE
INPUT FILE: AISC 360-10 - F.1-3B.STD
2. START JOB INFORMATION
3. ENGINEER DATE 25-OCT-12
4. JOB NAME AISC 360-10 VALIDATION
5. JOB CLIENT BENTLEY
6. JOB NO 7
7. END JOB INFORMATION
8. ********************************************************************************
9. * AISC EXAMPLE F.1-3
10. * W-SHAPE FLEXURAL MEMBER DESIGN IN STRONG-AXIS BENDING,
11. * BRACED AT MIDSPAN
12. * (1 ANALYTICAL MEMBER WITH 2 UNBRACED SEGMENT)
13. ********************************************************************************
14. INPUT WIDTH 79
15. UNIT INCHES KIP
16. JOINT COORDINATES
17. 1 0 0 0; 2 420 0 0
18. MEMBER INCIDENCES
19. 1 1 2
20. ********************************************************************************
21. DEFINE MATERIAL START
22. ISOTROPIC STEEL
23. E 29000
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50

CONSTANTS
MATERIAL STEEL ALL

SUPPORTS
STAAD SPACE

LOAD 1 LOADTYPE DEAD  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.0375

LOAD 2 LOADTYPE LIVE  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.0625

LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6

LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0

PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       3
TOTAL LOAD COMBINATION CASES =     2  SO FAR.
**Verification Examples**

**V.09 Steel Design**

---

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>71. UNB 210 ALL</td>
<td>-- PAGE NO.</td>
</tr>
<tr>
<td>72. *</td>
<td>-- PAGE NO.</td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>MAIN 1 ALL</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>75. TRACK 2 ALL</td>
<td>-- PAGE NO.</td>
</tr>
<tr>
<td>76. CHECK CODE ALL</td>
<td>-- PAGE NO.</td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>STEEL DESIGN</th>
<th>-- PAGE NO.</th>
</tr>
</thead>
</table>

---

### STEEL DESIGN

**STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a**

---

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE NOTED)**

---

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ FX</th>
<th>CRITICAL COND/ MY</th>
<th>RATIO/ MZ</th>
<th>LOADING/ LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST</td>
<td>W18X50</td>
<td>PASS</td>
<td>Eq. H1-1b 0.924</td>
<td>3</td>
<td>210.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>-3197.25</td>
<td>210.00</td>
</tr>
</tbody>
</table>

---

**SLENDERNESS**

- Actual Slenderness Ratio : 254.294
- Allowable Slenderness Ratio : 300.000

---

**STRENGTH CHECKS**

- Critical L/C : 3
- Ratio : 0.924 (PASS)
- Location : 210.00
- Condition : Eq. H1-1b

---

**DESIGN FORCES**

- $F_x$: 0.000E+00
- $F_y$: 0.000E+00
- $F_z$: 0.000E+00
- $M_x$: 0.000E+00
- $M_y$: 0.000E+00
- $M_z$: -3.197E+03

---

**SECTION PROPERTIES (UNIT: INCH)**

- $A_{zz}$: 8.550E+00
- $A_{yy}$: 6.390E+00
- $C_w$: 3.046E+03
- $S_{zz}$: 8.889E+01
- $S_{yy}$: 1.069E+01
- $I_{zz}$: 8.000E+02
- $I_{yy}$: 4.010E+01
- $I_x$: 1.240E+00

---

**MATERIAL PROPERTIES**

- $F_yld$: 50.000
- $F_u$: 65.000

---

**Actual Member Length**: 420.000

---

**Kz**: 1.00

---

**SECTION CLASS**

- UNSTIFFENED / l l p l r CASE

---

**STIFFENED**

- Compression : Non-Slender
  - 6.58
  - 13.49
  - T.B4.1(a)-1
- Slender
  - 45.23
  - 35.88
  - T.B4.1(a)-5
- Flexure : Compact
  - 6.58
  - 24.08
  - T.B4.1(b)-10
  - 45.23
  - 137.27
  - T.B4.1(b)-15

---

**STAAD SPACE**

---

**STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD) v1.4a**

---

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE NOTED)**

---
### CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>6.61E+02</td>
<td>0.00</td>
<td>Eq. D2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Rupture</td>
<td>7.17E+02</td>
<td>0.00</td>
<td>Eq. D2-2</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>5.09E+02</td>
<td>0.00</td>
<td>Eq. E7-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Min Buck</td>
<td>5.14E+01</td>
<td>0.00</td>
<td>Eq. E7-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Flexural</td>
<td>3.92E+02</td>
<td>0.00</td>
<td>Eq. E7-1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>2.31E+02</td>
<td>0.00</td>
<td>Eq. G2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>1.92E+02</td>
<td>0.159</td>
<td>Eq. G2-1</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>4.54E+03</td>
<td>0.703</td>
<td>Eq. F2-1</td>
<td>3</td>
<td>210.00</td>
</tr>
<tr>
<td>Minor</td>
<td>7.47E+02</td>
<td>0.00</td>
<td>Eq. F6-1</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>5.05E+03</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Intermediate My</td>
<td>8.30E+02</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>3.46E+03</td>
<td>0.924</td>
<td>Eq. F2-3</td>
<td>3</td>
<td>210.00</td>
</tr>
<tr>
<td>Intermediate Mn</td>
<td>3.84E+03</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Intermediate Me</td>
<td>3.84E+03</td>
<td>0.00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.924</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.924</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate Mcx / Mrx / Pc / Mcy</td>
<td>3.46E+03</td>
<td>3</td>
<td>210.00</td>
</tr>
</tbody>
</table>

---

**Verification Examples**

**V.09 Steel Design**

**STAAD.Pro** 3966  **User Manual**
7.47E+02  0.00E+00  0.00E+00
Flexure Tens  3.46E+03  3.20E+03  6.61E+02
7.47E+02  0.00E+00  0.00E+00

77. ***************************************************
78. LOAD LIST 4
79. PARAMETER 2
80. CODE AISC UNIFIED 2010
81. METHOD ASD
82. *
83. * ASTM A992
84. FYLD 50 ALL
85. FU 65 ALL
86. *CB 0 ALL
87. CB 1.3 ALL
88. *
89. LX 210 ALL
90. UNT 210 ALL
91. UNB 210 ALL
92. *
93. TRACK 2 ALL
94. CHECK CODE ALL
STEEL DESIGN
STAAD SPACE

--- PAGE NO. 7 ---

STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD) v1.4a
******************************************************************************

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
</tr>
<tr>
<td>1ST</td>
<td>W18X50</td>
<td>PASS</td>
</tr>
</tbody>
</table>

--- SLENDERNESS ---
Actual Slenderness Ratio :  254.294  L/C :   4
Allowable Slenderness Ratio :  300.000  LOC :  0.00

--- STRENGTH CHECKS ---
Critical L/C :  4  Ratio :  0.958(PASS)
Loc :  210.00  Condition :  Eq. H1-1b

--- DESIGN FORCES ---
Fx:  0.000E+00(F)  Fy:  0.000E+00  Fz:  0.000E+00
Mx:  0.000E+00  My:  0.000E+00  Mz:  -2.205E+03

--- SECTION PROPERTIES (UNIT: INCH) ---
Azz:  8.550E+00  Ayy:  6.390E+00  Cw:  3.046E+03
Szz:  8.889E+01  Syy:  1.069E+01
Izz:  8.000E+02  Iyy:  4.010E+01  Ix:  1.240E+00

--- MATERIAL PROPERTIES ---
Fyld:  50.000  Fu:  65.000

Actual Member Length:  420.000
Design Parameters
Kz:  1.00  Ky:  1.00  NSF:  1.00  SLF:  1.00  CSP:  12.00
### Section Class

<table>
<thead>
<tr>
<th>STIFFENED</th>
<th>UNSTIFFENED</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Compression</strong>: Non-Slender</td>
<td>6.58</td>
</tr>
<tr>
<td>Slender</td>
<td>45.23</td>
</tr>
<tr>
<td><strong>Flexure</strong>: Compact</td>
<td>6.58</td>
</tr>
<tr>
<td>Compact</td>
<td>45.23</td>
</tr>
</tbody>
</table>

### STAAD SPACE

--- PAGE NO.

**8**

---

**STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD) v1.4a**

---

**ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)**

### Check for Axial Tension

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>4.40E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>4</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>4.78E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

### Check for Axial Compression

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>3.39E+02</td>
<td>0.000</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>3.42E+01</td>
<td>0.000</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
</tbody>
</table>

### Flexural

| Tor Buck | 0.00E+00 | 2.61E+02 | 0.000 | Eq. E7-1 | 4 | 0.00 |

### Intermediate

<table>
<thead>
<tr>
<th>Results</th>
<th>Eff Area</th>
<th>KL/r</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>1.42E+01</td>
<td>56.93</td>
<td>3.85E+01</td>
<td>8.83E+01</td>
<td>5.65E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>1.47E+01</td>
<td>254.29</td>
<td>3.88E+00</td>
<td>4.43E+00</td>
<td>5.71E+01</td>
</tr>
</tbody>
</table>

### Check for Shear

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>1.54E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
<tr>
<td>Local-Y</td>
<td>2.10E+01</td>
<td>1.28E+02</td>
<td>0.164</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
</tbody>
</table>

### Intermediate

<table>
<thead>
<tr>
<th>Results</th>
<th>Aw</th>
<th>Cv</th>
<th>Kv</th>
<th>h/tw</th>
<th>Vn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>8.55E+00</td>
<td>1.00</td>
<td>1.20</td>
<td>6.58</td>
<td>2.56E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>6.39E+00</td>
<td>1.00</td>
<td>0.00</td>
<td>45.23</td>
<td>1.92E+02</td>
</tr>
</tbody>
</table>

### Check for Bending-Yielding

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>2.20E+03</td>
<td>3.02E+03</td>
<td>0.729</td>
<td>Eq. F2-1</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>4.97E+02</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>4</td>
</tr>
</tbody>
</table>

### Intermediate

<table>
<thead>
<tr>
<th>Mn</th>
<th>My</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>5.05E+03</td>
</tr>
<tr>
<td>Minor</td>
<td>8.30E+02</td>
</tr>
</tbody>
</table>

### Check for Bending-Lateral Torsional Buckling

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>2.20E+03</td>
<td>2.30E+03</td>
<td>0.958</td>
<td>Eq. F2-3</td>
<td>4</td>
</tr>
</tbody>
</table>

### Intermediate

| Mn | Me | Cb | Lp | Lr | Lb |
V. AISC 360-2005

V. AISC 360-05 Bending

To verify the flexural member strength in strong-axis bending of an I section for both LRFD and ASD methods per the AISC 360-05 code.

References

Details
Verify the Flexural Member Strength in Strong-Axis Bending by both LRFD and ASD method with ASTM A992 W18X50 beam with a simple span of 35 feet. The nominal loads are a uniform dead load of 0.45 kip/ft and a uniform live load of 0.75 kip/ft. Assuming the beam is continuously braced.

Validation
ASTM A992: $F_y = 50 \text{ ksi}, F_u = 65 \text{ ksi}$
Calculate the required flexural strength:

**LRFD**

\[
w_u = 1.2(0.450 \text{ kip/ft}) + 1.6(0.750 \text{ kip/ft}) = 1.74 \text{ kip/ft}
\]

\[
M_u = \frac{1.74 \text{ kip/ft} (35.0 \text{ ft}^2)}{8} = 266 \text{ kip-ft}
\]

**ASD**

\[
w_a = 0.450 \text{ kip/ft} + 0.750 \text{ kip/ft} = 1.20 \text{ kip/ft}
\]

\[
M_a = \frac{1.20 \text{ kip/ft} (35.0 \text{ ft}^2)}{8} = 184 \text{ kip-ft}
\]

Per the User Note in section F-2 of ref 2 (p.16.1-47), the W18X50 section is compact. Since the beam is continuously braced and compact, only the yielding limit state applies.

**Tip:** Per Table 3-2 of ref. 2 (p.3-17), the W18X50 is not the most efficient shape for this individual beam. This example follows that given in ref 1. STAAD.Pro will select a lighter beam if the member selection command is used.

**LRFD**

\[
\phi_b M_n = 0.9 \times (101 \text{ in}^3 \times 50 \text{ ksi})(1 \text{ ft} /12 \text{ in}) = 379 \text{ ft kip} > 266 \text{ ft kip}
\]

**ASD**

\[
M_n/\Omega_b = M_{px}/\Omega_b = (101 \text{ in}^3 \times 50 \text{ ksi})(1 \text{ ft} /12 \text{ in}) / 1.67 = 252 \text{ ft kip} > 180 \text{ ft kip}
\]

**Results**

**Table 550: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\phi_b M_n) (kips) [LRFD]</td>
<td>379</td>
<td>379</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>(M_n/\Omega_b) (kips) [ASD]</td>
<td>252</td>
<td>252</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2005\AISC 360-05 Bending.STD is typically installed with the program.
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI Y -0.45
LOAD 2 LIVE LOAD
MEMBER LOAD
1 UNI Y -0.75
LOAD COMB 3 SERVICE LOAD
1 1.0 2 1.0
LOAD COMB 4 ULTIMATE LOAD
1 1.2 2 1.6
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ LOAD</th>
<th>CRITICAL COND/ LOAD</th>
<th>RATIO/ LOAD</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>* 1 ST W18X50</td>
<td>(AISC SECTIONS)</td>
<td>FAIL</td>
<td>Eq. H1-1b</td>
<td>3.331</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>-183.75</td>
</tr>
</tbody>
</table>

SLENDERNESS
Actual Slenderness Ratio : 254.294 L/C : 3
Allowable Slenderness Ratio : 300.000 LOC : 0.00
**STRENGTH CHECKS**

Critical L/C : 3   Ratio : 3.331 (FAIL)  
Loc : 17.50  Condition : Eq. H1-1b

**DESIGN FORCES**

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>4.40E+02</td>
<td>0.00</td>
<td>Eq. D2-1</td>
<td>3</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>4.78E+02</td>
<td>0.00</td>
<td>Eq. D2-2</td>
<td>3</td>
</tr>
</tbody>
</table>

**SECTION PROPERTIES (UNIT: INCH)**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Azz</td>
<td>8.550E+00</td>
</tr>
<tr>
<td>Ayy</td>
<td>6.390E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>3.046E+03</td>
</tr>
<tr>
<td>Szz</td>
<td>8.889E+01</td>
</tr>
<tr>
<td>Syy</td>
<td>1.069E+01</td>
</tr>
<tr>
<td>Izz</td>
<td>8.000E+02</td>
</tr>
<tr>
<td>Iyy</td>
<td>4.010E+01</td>
</tr>
<tr>
<td>Ix</td>
<td>1.240E+00</td>
</tr>
</tbody>
</table>

**MATERIAL PROPERTIES**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld</td>
<td>7199.999</td>
</tr>
<tr>
<td>Fu</td>
<td>9359.999</td>
</tr>
</tbody>
</table>

Actual Member Length: 35.000

Design Parameters:

Kz: 1.00  Ky: 1.00  NSF: 1.00  SLF: 1.00  CSP: 12.00

**SECTION CLASS**

<table>
<thead>
<tr>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>STIFFENED</td>
<td></td>
</tr>
</tbody>
</table>

**STAAAD PLANE**

CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>4.40E+02</td>
<td>0.00</td>
<td>Eq. D2-1</td>
<td>3</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>4.78E+02</td>
<td>0.00</td>
<td>Eq. D2-2</td>
<td>3</td>
</tr>
</tbody>
</table>

CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>3.39E+02</td>
<td>0.00</td>
<td>Sec. E1</td>
<td>3</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>3.42E+01</td>
<td>0.00</td>
<td>Eq. E7-1</td>
<td>3</td>
</tr>
<tr>
<td>Tor Buck</td>
<td>0.00E+00</td>
<td>1.73E+02</td>
<td>0.00</td>
<td>Eq. E7-1</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate Results

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>9.88E-02</td>
<td>56.93</td>
<td>5.54E+03</td>
<td>1.27E+04</td>
<td>5.65E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>1.02E-01</td>
<td>254.29</td>
<td>5.59E+02</td>
<td>6.37E+02</td>
<td>5.71E+01</td>
</tr>
<tr>
<td>Flexural</td>
<td>1.02E-01</td>
<td>2.82E+03</td>
<td>2.88E+02</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>1.54E+02</td>
<td>0.00</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
</tbody>
</table>
### Local-Y

<p>| | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Local-Z</td>
<td>5.94E-02</td>
<td>1.00</td>
<td>1.20</td>
<td>6.58</td>
<td>2.56E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>4.44E-02</td>
<td>1.00</td>
<td>0.00</td>
<td>45.23</td>
<td>1.92E+02</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>1.84E+02</td>
<td>2.52E+02</td>
<td>0.729</td>
<td>Eq. F2-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>4.14E+01</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>3</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>1.84E+02</td>
<td>5.52E+01</td>
<td>3.331</td>
<td>Eq. F2-3</td>
<td>3</td>
</tr>
</tbody>
</table>

### CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>3.331</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>3.331</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro Code Checking - (AISC-360-05-ASD) v3.3a**

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)**

### Member Table

<table>
<thead>
<tr>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/</th>
<th>RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FX MY MZ LOCATION</td>
<td></td>
</tr>
<tr>
<td>*</td>
<td>1 ST W18X50</td>
<td></td>
</tr>
</tbody>
</table>

**Verification Examples**

**STAAD.Pro User Manual**
SLENDERNESS
Actual Slenderness Ratio :  254.294  L/C :   4
Allowable Slenderness Ratio :  300.000  LOC :  0.00

STRENGTH CHECKS
Critical L/C :    4  Ratio :  3.213(FAIL)
Loc :  17.50  Condition :  Eq. H1-1b

DESIGN FORCES
Fx:  0.000E+00( ) Fy:  0.000E+00  Fz:  0.000E+00
Mx:  0.000E+00  My:  0.000E+00  Mz:  -2.664E+02

SECTION PROPERTIES (UNIT: INCH)
Azz:  8.550E+00  Ayy:  6.390E+00  Cw:  3.046E+03
Szz:  8.889E+01  Syy:  1.069E+01
Izz:  8.000E+02  Iyy:  4.010E+01  Ix:  1.240E+00

MATERIAL PROPERTIES
Fyld:  7199.999  Fu:  9359.999

Actual Member Length:  35.000
Design Parameters
Kz:  1.00  Ky:  1.00  NSF:  1.00  SLF:  1.00  CSP:  12.00

SECTION CLASS
UNSTIFFENED /  l  l p  l r  CASE
STIFFENED
Compression :  Non-Slender  6.58  N/A  13.49  T.B4.1-3
Slender  45.23  N/A  35.88  T.B4.1-10
Flexure     :  Compact  6.58  9.15  24.08  T.B4.1-1
Compact  45.23  90.55  137.27  T.B4.1-9

STAAD PLANE
--- PAGE NO.

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)

CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>6.61E+02</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>4</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>7.17E+02</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>5.09E+02</td>
<td>0.000</td>
<td>Sec. E1</td>
<td>4</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>5.14E+01</td>
<td>0.000</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
<tr>
<td>Flexural</td>
<td>0.00E+00</td>
<td>2.59E+02</td>
<td>0.000</td>
<td>Eq. E7-1</td>
<td>4</td>
</tr>
</tbody>
</table>

Intermediate Results
Eff Area  KL/r  Fcr  Fe  Pn
Maj Buck  9.88E-02  56.93  5.54E+03  1.27E+04  5.65E+02
Min Buck  1.02E-01  254.29  5.59E+02  6.37E+02  5.71E+01
Flexural  Ag  Fcr  Pn
Tor Buck  1.02E-01  2.82E+03  2.88E+02
V. AISC 360-05 Compression

Verify the axial compression capacity for a the y-y axis of an I section for both the LRFD and ASD methods per the AISC 360-05 code.

References

1. Design Examples, Version 13.0, American Institute of Steel Construction, 2005, page E-4. Example E.1a

Details

The column is 30 ft long, pinned top and bottom in both axis.
The section is an ASTM A992 (Fy = 50 ksi) W-shape column to carry an axial dead load of 140 kips and live load of 420 kips. The column is 30 feet long.

Validation

Calculate the required strength.

LRFD:

\[ P_u = 1.2(140 \text{ kips}) + 1.6(420 \text{ kips}) = 840 \text{ kips} \]

ASD:

\[ P_a = 140 \text{ kips} + 420 \text{ kips} = 560 \text{ kips} \]

For a pinned-pinned condition, \( K = 1.0 \) (Table C-C2.2 of ref. 2). Because the unbraced length is equal about both x-x and y-y directions, y-y axis buckling governs.

Use Table 4-1 in ref 2 and select a W14x132 (\( K_{Ly} = 30 \text{ ft} \)). From this table:

\[ \phi_c P_n = 892 \text{ kips} > 840 \text{ kips} \]

\[ P_n/\Omega_c = 594 \text{ kips} > 560 \text{ kips} \]

Results

**Table 551: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \phi_c P_n ) (kips) [LRFD]</td>
<td>892</td>
<td>893</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>( P_n/\Omega_c ) (kips) [ASD]</td>
<td>594</td>
<td>594</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2005\AISC 360-05 Compression.STD is typically installed with the program.

STAAD PLANE

START JOB INFORMATION

ENGINEER DATE 29-Aug-18

END JOB INFORMATION

INPUT WIDTH 79

UNIT FEET KIP

JOINT COORDINATES

1 0 0 0; 2 0 30 0;

MEMBER INCIDENCES

1 1 2;

DEFINE MATERIAL START

ISOTROPIC STEEL

E 4.176e+06

POISSON 0.3

DENSITY 0.489024

ALPHA 6.5e-06

DAMP 0.03

ISOTROPIC STEEL_50_KSI

E 4.176e+06

POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MZ
LOAD 1 DEAD LOAD
JOINT LOAD
2 FY -140
LOAD 2 LIVE LOAD
JOINT LOAD
2 FY -420
LOAD COMB 3 SERVICE LOAD
1 1.0 2 1.0
LOAD COMB 4 ULTIMATE LOAD
1 1.2 2 1.6
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/CRITICAL/</th>
<th>RATIO/LOADING/</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX   MY   MZ</td>
<td>0.943  3</td>
<td></td>
</tr>
<tr>
<td>1 ST</td>
<td>W14X132</td>
<td>560.00 C</td>
<td>0.00  0.00  0.00</td>
<td></td>
</tr>
</tbody>
</table>

SLENDERNESS
Actual Slenderness Ratio : 95.792  L/C : 3
Allowable Slenderness Ratio : 200.000  LOC : 0.00

Verification Examples
V.09 Steel Design
### STRENGTH CHECKS

Critical L/C : 3  Ratio : 0.943 (PASS)

Loc : 0.00  Condition : Eq. H1-1a

### DESIGN FORCES

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fx</td>
<td>5.60E+02</td>
</tr>
<tr>
<td>Fy</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Fz</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Mx</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>My</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Mz</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

### SECTION PROPERTIES (UNIT: INCH)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Azz</td>
<td>3.028E+01</td>
</tr>
<tr>
<td>Ayy</td>
<td>9.481E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>2.560E+04</td>
</tr>
<tr>
<td>Szz</td>
<td>2.082E+02</td>
</tr>
<tr>
<td>Syy</td>
<td>7.456E+01</td>
</tr>
<tr>
<td>Izz</td>
<td>1.530E+03</td>
</tr>
<tr>
<td>Iyy</td>
<td>5.480E+02</td>
</tr>
<tr>
<td>Ix</td>
<td>1.230E+01</td>
</tr>
</tbody>
</table>

### MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld</td>
<td>50.000</td>
</tr>
<tr>
<td>Fu</td>
<td>60.000</td>
</tr>
</tbody>
</table>

### Actual Member Length: 360.000

### Design Parameters

Kz: 1.00  Ky: 1.00  NSF: 1.00  SLF: 1.00  CSP: 12.00

### CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

### CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>5.60E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>5.60E+02</td>
</tr>
<tr>
<td>Flexural</td>
<td>5.60E+02</td>
</tr>
<tr>
<td>Tor Buck</td>
<td>5.60E+02</td>
</tr>
<tr>
<td>Intermediate</td>
<td></td>
</tr>
</tbody>
</table>

### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro Code Checking - (AISC-360-05-ASD) v3.3a**

**Verification Examples**

V.09 Steel Design

---

**STAAD Pro 3978 User Manual**
### Intermediate

<table>
<thead>
<tr>
<th>Results</th>
<th>$A_w$</th>
<th>$C_v$</th>
<th>$K_v$</th>
<th>$h/tw$</th>
<th>$V_n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>3.03E+01</td>
<td>1.00</td>
<td>1.20</td>
<td>7.14</td>
<td>9.08E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>9.48E+00</td>
<td>1.00</td>
<td>0.00</td>
<td>17.74</td>
<td>2.84E+02</td>
</tr>
</tbody>
</table>

---

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>7.01E+03</td>
<td>0.000</td>
<td>Eq. F2-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>3.38E+03</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>3</td>
</tr>
</tbody>
</table>

---

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>5.97E+03</td>
<td>0.000</td>
<td>Eq. F2-2</td>
<td>3</td>
</tr>
</tbody>
</table>

---

### CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.943</td>
<td>Eq. H1-1a</td>
<td>3</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.000</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
</tbody>
</table>

---

### SLENDERNESS Verification Examples

V.09 Steel Design

STAAD.Pro User Manual
<table>
<thead>
<tr>
<th>Actual Slenderness Ratio : 95.792</th>
<th>L/C : 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable Slenderness Ratio : 200.000</td>
<td>LOC : 0.00</td>
</tr>
</tbody>
</table>

---

**STRENGTH CHECKS**

Critical L/C : 4 Ratio : 0.941 (PASS)

Loc : 0.00 Condition : Eq. H1-1a

---

**DESIGN FORCES**

Fx: 8.400E+02 (C) Fy: 0.000E+00 Fz: 0.000E+00

Mx: 0.000E+00 My: 0.000E+00 Mz: 0.000E+00

---

**SECTION PROPERTIES (UNIT: INCH)**

Azz: 3.028E+01 Ayy: 9.481E+00 Cw: 2.560E+04

Szz: 2.082E+02 Syy: 7.456E+01

Izz: 1.530E+03 Iyy: 5.480E+02 Ix: 1.230E+01

---

**MATERIAL PROPERTIES**

Fyld: 50.000 Fu: 60.000

---

<table>
<thead>
<tr>
<th>Actual Member Length: 360.000</th>
</tr>
</thead>
</table>

**DESIGN PARAMETERS**

Kz: 1.00 Ky: 1.00 NSF: 1.00 SLF: 1.00 CSP: 12.00

---

<table>
<thead>
<tr>
<th>SECTION CLASS</th>
<th>UNSTIFFENED /</th>
<th>1</th>
<th>1</th>
<th>p</th>
<th>1</th>
<th>r</th>
<th>CASE</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>STIFFENED</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**CHECK FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>0.00E+00</td>
<td>1.75E+03</td>
<td>0.000</td>
<td>Eq. D2-1</td>
<td>4</td>
</tr>
<tr>
<td>Rupture</td>
<td>0.00E+00</td>
<td>1.75E+03</td>
<td>0.000</td>
<td>Eq. D2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

---

**CHECK FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>8.40E+02</td>
<td>1.37E+03</td>
<td>0.612</td>
<td>Sec. E1</td>
<td>4</td>
</tr>
<tr>
<td>Min Buck</td>
<td>8.40E+02</td>
<td>8.93E+02</td>
<td>0.941</td>
<td>Eq. E3-1</td>
<td>4</td>
</tr>
</tbody>
</table>

---

**CHECK FOR SHEAR**

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>3.88E+01</td>
<td>57.33</td>
<td>3.93E+01</td>
<td>8.71E+01</td>
<td>1.53E+03</td>
</tr>
<tr>
<td>Min Buck</td>
<td>3.88E+01</td>
<td>95.79</td>
<td>2.56E+01</td>
<td>3.12E+01</td>
<td>9.92E+02</td>
</tr>
</tbody>
</table>

---

**STAAD PRO CODE CHECKING - (AISC-360-05-LRFD) v3.3a**

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE NOTED)
### Check for Bending-Yielding

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>1.05E+04</td>
<td>0.000</td>
<td>Eq. F2-1</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>5.08E+03</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>4</td>
</tr>
</tbody>
</table>

### Check for Bending-Lateral Torsional Buckling

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>8.98E+03</td>
<td>0.000</td>
<td>Eq. F2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

### Check for Flexure Tens/Comp Interaction

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.941</td>
<td>Eq. H1-1a</td>
<td>4</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.000</td>
<td>Eq. H1-1b</td>
<td>4</td>
</tr>
</tbody>
</table>

### V. AISC 360-05 Tension

Verify the tensile yield strength and tensile rupture strength of an I section using both the LRFD and ASD methods per the AISC 360-05 code.

Reference

1. Design Examples, Version 13.0, American Institute of Steel Construction, 2005, page D-2. Example D.1

Details

Verify the member Tensile Yield strength and Tensile Rupture Strength by both LRFD and ASD with the bolted end connection of a I Section (W8X21) using the LRFD and ASD method

Steel Gr. ASTM A992.
The connection consists of 2 rows of 4-3/4"Ø bolts in each flange (16 bolts total) in standard holes. The holes are spaced evenly at 3" on center, with a 1-1/4" edge to center at the end of the flange in line with the tensile force.

**Validation**

ASTM A992: $F_y = 50$ ksi, $F_u = 65$ ksi

Assume a W8X21 section:

- $A_g = 6.16$ in$^2$
- $b_f = 5.27$ in
- $t_f = 0.400$ in
- $d = 8.28$ in
- $r_y = 1.26$ in
- $y = 0.831$ in (for a WT4X10.5)

Calculate the required tensile strength.

**LRFD**

$$P_u = 1.2(30 \text{ kips}) + 1.6(90 \text{ kips}) = 180 \text{ kips}$$

**ASD**

$$P_a = 30 \text{ kips} + 90 \text{ kips} = 120 \text{ kips}$$

Check the tensile yield strength limit state using tabulated values per Table 5-1 in ref. 2:

$$\phi_t P_n = 277 \text{ kips} > 180 \text{ kips}$$

$$P_n/\Omega_t = 184 \text{ kips} > 120 \text{ kips}$$

Check the available tensile rupture strength at the end connection. The tabulated values are for $A_e/A_g \geq 0.75$. Verify this assumption is applicable.

Calculate $U$ as the larger of the values from Table D3.1 case 2 or case 7 (ref. 2).

**Case 2** - Check as two WT shapes:

$$U = 1 - x/l = 1 - 0.831 \text{ in} / 9.0 \text{ in} = 0.908$$

where

- $l =$ length of the bolted connection = 3x 3.0 in = 9.0 in
- $x =$ distance from the flange face to the plastic neutral axis. This is taken from Table 1-8 of ref. 2.

**Case 7**

$$b_f = 5.27 \text{ in} < 2/3d = 2/3(8.28 \text{ in}) = 5.52 \text{ in}; U = 0.85$$

Use $U = 0.908$

Calculate the net area at the bolted connection:

$$A_n = A_g - 4(\phi_h + 1/16 \text{ in})t_f = 6.16 \text{ in}^2 - 4(13/16 \text{ in} + 1/16 \text{ in})(0.400 \text{ in}) = 4.76 \text{ in}^2$$

Calculate the effective area:

$$A_e = A_n U = 4.76 \text{ in}^2 \times 0.908 = 4.32 \text{ in}^2$$

$$A_e/A_g = 4.32 \text{ in}^2 / 6.16 \text{ in}^2 = 0.701 < 0.75$$
Therefore, the tabulated values for rupture do not apply. Calculate the rupture strength:

\[ P_n = F_u A_e = 65 \text{ ksi (4.32 in}^2) = 281 \text{ kips} \]

LRFD

\[ \phi P_n = 0.75(281 \text{ kips}) = 211 \text{ kips} > 180 \text{ kips} \]

ASD

\[ P_n/\Omega_t = 281 \text{ kips}/2.00 = 141 \text{ kips} > 120 \text{ kips} \]

Results

Table 552: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\phi P_n) (kips) [LRFD]</td>
<td>Yield</td>
<td>277</td>
<td>277</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td>Rupture</td>
<td>211</td>
<td>211</td>
<td>none</td>
</tr>
<tr>
<td>(P_n/\Omega_t) (kips) [ASD]</td>
<td>Yield</td>
<td>184</td>
<td>184</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td>Rupture</td>
<td>141</td>
<td>141</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2005\AISC 360-05 Tension.STD is typically installed with the program.

STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 27-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 25 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X21
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
2 PINNED
1 FIXED BUT FY MZ
LOAD 1 DEAD LOAD
JOINT LOAD
1 FY -30
LOAD 2 LIVE LOAD
JOINT LOAD
1 FY -90
LOAD COMB 3 SERVICE LOAD
1 1.0 2 1.0
LOAD COMB 4 ULTIMATE LOAD
1 1.2 2 1.6
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
NSF 0.773 ALL
SLF 0.908 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD LRFD
TRACK 2 ALL
NSF 0.773 ALL
SLF 0.908 ALL
FU 9360 ALL
FYLD 7200 ALL
CHECK CODE ALL
FINISH

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT</th>
<th>CRITICAL CONDITION</th>
<th>RATIO</th>
<th>LOADING</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PASS</td>
<td>Eq. H1-1a</td>
<td>0.854</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>120.00</td>
<td>T</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>SLENDERNESS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Actual Slenderness Ratio</td>
<td>238.212</td>
<td>L/C</td>
<td>3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Allowable Slenderness Ratio</td>
<td>300.000</td>
<td>LOC</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Output**
Critical L/C : 3 Ratio : 0.854 (PASS)

Location : 0.00 Condition : Eq. H1-1a

Design Forces
Fx: 1.200E+02 (T) Fy: 0.000E+00 Fz: 0.000E+00
Mx: 0.000E+00 My: 0.000E+00 Mz: 0.000E+00

Section Properties (Unit: Inch)
Azz: 4.216E+00 Ayy: 2.070E+00 Cw: 1.517E+02
Szz: 1.819E+01 Syy: 3.708E+00
Izz: 7.530E+01 Iyy: 9.770E+00 Ix: 2.820E-01

Material Properties
Fyld: 7199.999 Fu: 9359.999

Actual Member Length: 25.000

Design Parameters
Kz: 1.00 Ky: 1.00 NSF: 0.77 SLF: 0.91 CSP: 12.00

Section Class
Unstiffened / l l p l r CASE

Stiffened
Compression: Non-Slender 6.59 N/A 13.49 T.B4.1-3
Non-Slender 27.52 N/A 35.88 T.B4.1-10
Flexure: Compact 6.59 9.15 24.08 T.B4.1-1
Compact 27.52 90.55 137.27 T.B4.1-9

STAAD Plane

--- Page No.

STAAD Pro Code Checking - (AISC-360-05-ASD) v3.3a
All Units Are - Kip Feet (Unless Otherwise Noted)

Check for Axial Tension

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>1.20E+02</td>
<td>1.84E+02</td>
<td>0.651</td>
<td>Eq. D2-1</td>
<td>3</td>
</tr>
<tr>
<td>Rupture</td>
<td>1.20E+02</td>
<td>1.41E+02</td>
<td>0.854</td>
<td>Eq. D2-2</td>
<td>3</td>
</tr>
</tbody>
</table>

Check for Axial Compression

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>1.88E+02</td>
<td>0.000</td>
<td>Sec. E1</td>
<td>3</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>1.63E+01</td>
<td>0.000</td>
<td>Eq. E3-1</td>
<td>3</td>
</tr>
<tr>
<td>Flexural Tor Buck</td>
<td>0.00E+00</td>
<td>1.13E+02</td>
<td>0.000</td>
<td>Eq. E4-1</td>
<td>3</td>
</tr>
<tr>
<td>Intermediate Results</td>
<td>Eff Area</td>
<td>KL/r</td>
<td>Fcr</td>
<td>Fe</td>
<td>Pn</td>
</tr>
<tr>
<td>Maj Buck</td>
<td>4.28E-02</td>
<td>85.81</td>
<td>4.20E+03</td>
<td>5.60E+03</td>
<td>1.80E+02</td>
</tr>
<tr>
<td>Min Buck</td>
<td>4.28E-02</td>
<td>238.21</td>
<td>6.37E+02</td>
<td>7.26E+02</td>
<td>2.72E+01</td>
</tr>
<tr>
<td>Flexural</td>
<td>Ag</td>
<td>Fcr</td>
<td>Pn</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tor Buck</td>
<td>4.28E-02</td>
<td>4.41E+03</td>
<td>1.89E+02</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Check for Shear

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>7.57E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>4.14E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>3</td>
</tr>
<tr>
<td>Results</td>
<td>Aw</td>
<td>Cv</td>
<td>Kv</td>
<td>h/tw</td>
<td>Vn</td>
</tr>
<tr>
<td>---------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
<td>--------</td>
<td>------------------</td>
</tr>
<tr>
<td>Local-Z</td>
<td>2.93E-02</td>
<td>1.00</td>
<td>1.20</td>
<td>6.59</td>
<td>1.26E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>1.44E-02</td>
<td>1.00</td>
<td>0.00</td>
<td>27.52</td>
<td>6.21E+01</td>
</tr>
</tbody>
</table>

**CHECK FOR BENDING-YIELDING**

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>5.09E+01</td>
<td>0.000</td>
<td>Eq. F2-1</td>
<td>3</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>1.42E+01</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate | Mn | My |
| Major | 8.50E+01 | 0.00E+00 | |
| Minor | 2.37E+01 | 0.00E+00 | |

**CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING**

<table>
<thead>
<tr>
<th>Force</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>1.68E+01</td>
<td>0.000</td>
<td>Eq. F2-3</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate | Mn | Me | Cb | Lp | Lr | Lb |
| Major | 2.80E+01 | 0.00E+00 | 1.00 | 4.45 | 14.75 | 25.00 |

**CHECK FOR FLEXURE TENS/CMP INTERACTION**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.000</td>
<td>Eq. H1-1b</td>
<td>3</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.854</td>
<td>Eq. H1-1a</td>
<td>3</td>
</tr>
</tbody>
</table>

Intermediate | Mx | My | Px |
| Flexure Comp | 1.68E+01 | 0.00E+00 | 1.63E+01 |
| Flexure Tens | 1.42E+01 | 0.00E+00 | 1.20E+02 |

---

55. LOAD LIST 4
56. PARAMETER 2
57. CODE AISC UNIFIED 2005
58. METHOD LRFD
59. TRACK 2 ALL
60. NSF 0.773 ALL
61. SLF 0.908 ALL
62. FU 9360 ALL
63. FYLD 7200 ALL
64. CHECK CODE ALL

STEEL DESIGN

---

5. LOAD LIST 4
6. PARAMETER 2
7. CODE AISC UNIFIED 2005
8. METHOD LRFD
9. TRACK 2 ALL
10. NSF 0.773 ALL
11. SLF 0.908 ALL
12. FU 9360 ALL
13. FYLD 7200 ALL
14. CHECK CODE ALL

STEEL DESIGN

---
SLENDERNESS
Actual Slenderness Ratio : 238.212 L/C : 4
Allowable Slenderness Ratio : 300.000 LOC : 0.00

STRENGTH L/C : 4 Ratio : 0.854(PASS)
Loc : 0.00 Condition : Eq. H1-1a

DESIGN FORCES
Fx: 1.800E+02(T ) Fy: 0.000E+00 Fz: 0.000E+00
Mx: 0.000E+00 My: 0.000E+00 Mz: 0.000E+00

SECTION PROPERTIES (UNIT: INCH)
Azz: 4.216E+00 Ayy: 2.070E+00 Cw: 1.517E+2
Szz: 1.819E+01 Syy: 3.708E+00
Izz: 7.530E+01 Iyy: 9.770E+00 Ix: 2.820E-01

MATERIAL PROPERTIES
Fyld: 7199.999 Fu: 9359.999

Actual Member Length: 25.000
Design Parameters
Kz: 1.00 Ky: 1.00 NSF: 0.77 SLF: 0.91 CSP: 12.00

SECTION CLASS UNSTIFFENED / l l p l r CASE
STIFFENED
Compression : Non-Slender 6.59 N/A 13.49 T.B4.1-3
Non-Slender 27.52 N/A 35.88 T.B4.1-10
Flexure : Compact 6.59 9.15 24.08 T.B4.1-1
Compact 27.52 90.55 137.27 T.B4.1-9

STAAD PLANE -- PAGE NO.

--- PAGE NO. 7 ---

CHECK FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield</td>
<td>1.80E+02</td>
<td>2.77E+02</td>
<td>0.649</td>
<td>Eq. D2-1</td>
<td>4</td>
</tr>
<tr>
<td>Rupture</td>
<td>1.80E+02</td>
<td>2.11E+02</td>
<td>0.854</td>
<td>Eq. D2-2</td>
<td>4</td>
</tr>
</tbody>
</table>

CHECK FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>FORCE</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>0.00E+00</td>
<td>1.62E+02</td>
<td>0.000</td>
<td>Sec. E1</td>
<td>4</td>
</tr>
<tr>
<td>Min Buck</td>
<td>0.00E+00</td>
<td>2.45E+01</td>
<td>0.000</td>
<td>Eq. E3-1</td>
<td>4</td>
</tr>
<tr>
<td>Flexural</td>
<td>0.00E+00</td>
<td>1.70E+02</td>
<td>0.000</td>
<td>Eq. E4-1</td>
<td>4</td>
</tr>
</tbody>
</table>

Results

<table>
<thead>
<tr>
<th>Eff Area</th>
<th>KL/r</th>
<th>Fcr</th>
<th>Fe</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maj Buck</td>
<td>4.28E-02</td>
<td>85.81</td>
<td>4.20E+03</td>
<td>5.60E+03</td>
</tr>
<tr>
<td>Min Buck</td>
<td>4.28E-02</td>
<td>238.21</td>
<td>6.37E+02</td>
<td>7.26E+02</td>
</tr>
</tbody>
</table>

Flexural Tor Buck

<table>
<thead>
<tr>
<th>Ag</th>
<th>Fcr</th>
<th>Pn</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.28E-02</td>
<td>4.41E+03</td>
<td>1.89E+02</td>
</tr>
</tbody>
</table>
## Verification Examples

### V.09 Steel Design

### CHECK FOR SHEAR

<table>
<thead>
<tr>
<th>FORCES</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Z</td>
<td>0.00E+00</td>
<td>1.14E+02</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
<tr>
<td>Local-Y</td>
<td>0.00E+00</td>
<td>6.21E+01</td>
<td>0.000</td>
<td>Eq. G2-1</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate Results</td>
<td>A_w</td>
<td>C_v</td>
<td>K_v</td>
<td>h/t_w</td>
<td>V_m</td>
</tr>
<tr>
<td>Local-Z</td>
<td>2.93E-02</td>
<td>1.00</td>
<td>1.20</td>
<td>6.59</td>
<td>1.26E+02</td>
</tr>
<tr>
<td>Local-Y</td>
<td>1.44E-02</td>
<td>1.00</td>
<td>0.00</td>
<td>27.52</td>
<td>6.21E+01</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-YIELDING

<table>
<thead>
<tr>
<th>FORCES</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>7.65E+01</td>
<td>0.000</td>
<td>Eq. F2-1</td>
<td>4</td>
</tr>
<tr>
<td>Minor</td>
<td>0.00E+00</td>
<td>2.13E+01</td>
<td>0.000</td>
<td>Eq. F6-1</td>
<td>4</td>
</tr>
</tbody>
</table>

### CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING

<table>
<thead>
<tr>
<th>FORCES</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major</td>
<td>0.00E+00</td>
<td>2.52E+01</td>
<td>0.000</td>
<td>Eq. F2-3</td>
<td>4</td>
</tr>
</tbody>
</table>

### CHECK FOR FLEXURE TENS/COMP INTERACTION

<table>
<thead>
<tr>
<th>FORCES</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexure Comp</td>
<td>0.000</td>
<td>Eq. H1-1b</td>
<td>4</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>0.854</td>
<td>Eq. H1-1a</td>
<td>4</td>
</tr>
<tr>
<td>Intermediate Results</td>
<td>M_c_x /</td>
<td>M_r_x /</td>
<td>P_c /</td>
</tr>
<tr>
<td>Flexure Comp</td>
<td>2.52E+01</td>
<td>0.00E+00</td>
<td>2.45E+01</td>
</tr>
<tr>
<td>Flexure Tens</td>
<td>2.13E+01</td>
<td>0.00E+00</td>
<td>2.11E+02</td>
</tr>
</tbody>
</table>

---

### V. Steel Design per AISC ASD

**V. AISC ASD - Column Compression Capacity 1**

Determine the axial compression capacity of a W14X132 column.

**Reference**


**Problem**

\[ F_y = 36 \text{ ksi} \]
Comparison

The reported STAAD.Pro value is obtained by multiplying $FA$ times $AX$ from the STAAD.Pro output:

$$17.36 \text{ ksi (38.8 in}^2) = 673.6 \text{ kips}$$

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable load (kips)</td>
<td>679</td>
<td>673.6</td>
<td>0.8%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Column Compression Capacity 1.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.

<table>
<thead>
<tr>
<th>ENGINEER DATE 22-Sep-18</th>
</tr>
</thead>
<tbody>
<tr>
<td>END JOB INFORMATION</td>
</tr>
</tbody>
</table>

* PAGE 3-4 OF 9TH ED. AISC-ASD. EXAMPLE 1. FA SHOULD BE 679/38.8 = 0.8% |

UNIT FEET KIP

1 0 0 0; 2 0 31 0;

MEMBER INCIDENCES

1 1 2;

MEMBER PROPERTY AMERICAN

1 TABLE ST W14X132

UNIT INCHES KIP

DEFINE MATERIAL START

ISOTROPIC STEEL_50_KSI

E 29000

POISSON 0.3

DENSITY 0.000283

ALPHA 6.5e-06

DAMP 0.03

TYPE STEEL

STRENGTH FY 50 FU 62 RY 1.5 RT 1.2

END DEFINE MATERIAL

UNIT FEET KIP

CONSTANTS

MATERIAL STEEL_50_KSI ALL

SUPPORTS

1 FIXED

LOAD 1

JOINT LOAD

2 FY -670

PERFORM ANALYSIS

UNIT INCHES KIP

PARAMETER 1

CODE AISC

FYLD 36 ALL

LY 192 ALL
STAAD Output

<table>
<thead>
<tr>
<th>MEMBERS</th>
<th>DESIGN CODE</th>
<th>MATERIALS</th>
<th>AXIAL</th>
<th>Y-PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>AISC-1989</td>
<td>ST W14X132</td>
<td>38.80</td>
<td>AX = 38.80</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>8.48</td>
<td>AY = 8.48</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>20.33</td>
<td>AZ = 20.33</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>74.56</td>
<td>SY = 74.56</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>208.16</td>
<td>SZ = 208.16</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3.76</td>
<td>RY = 3.76</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>6.28</td>
<td>RZ = 6.28</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>LOCATION</th>
<th>LOADING</th>
<th>VALUE</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOMENT-Y</th>
<th>MOMENT-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>L0 L0 L0 L0 L0 L0 L0 L0 L0</td>
<td>0.0</td>
<td>1</td>
<td>670.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RESULT/</th>
<th>CRITICALCOND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
</tr>
<tr>
<td>PASS</td>
<td>AISC-H1-1</td>
<td>0.994</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>
V. AISC ASD - Column Compression Capacity 2

Determine the axial compression capacity of a W12X106 column.

Reference


Problem

\[
F_y = 36 \text{ ksi} \\
K_y L_y = 11 \text{ ft} \\
K_z L_z = 22 \text{ ft}
\]

Comparison

The reported STAAD.Pro value is obtained by multiplying \( FA \times AX \) from the STAAD.Pro output = 18.30(31.20) = 571.0.

**Table 553: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable load (kips)</td>
<td>577</td>
<td>571.0</td>
<td>1.0%</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Column Compression Capacity 2.STD is typically installed with the program.

**STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.**

START JOB INFORMATION

ENGINEER DATE 22-Sep-18

END JOB INFORMATION

* PAGE 3-4 OF 9TH ED. AISC-ASD. EXAMPLE 2. 
* RATIO SHOULD BE APPROXIMATELY 540/577 = 0.936

UNIT FEET KIP

JOINT COORDINATES

1 0 0 0; 2 0 11 0;

MEMBER INCIDENCES

1 1 2;

MEMBER PROPERTY AMERICAN

1 TABLE ST W12X106
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -540
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
TRACK 2 ALL
KZ 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY -- PAGE NO.

|---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
| KL/R-Y= 42.50  | L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA = 18.30 |
| KL/R-Z= 48.28  | L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fA = 17.31 |
| UNL = 132.00   | L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ = 23.76 |
| CB = 1.00      | L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ = 23.76 |
| CMY = 0.85     | L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY = 27.00 |
| CMZ = 0.85     | L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY = 27.00 |

Verification Examples
V.09 Steel Design

STAAD.Pro
3992
User Manual
V. AISC ASD - Column Compression Capacity 3

Determine the axial compression capacity of a W12X40 column.

Reference


Problem

\[ F_y = 36 \text{ ksi} \]
\[ K_y L_y = K_z L_z = 16\text{ft} \]

Comparison

The reported STAAD.Pro value is obtained by multiplying \( FA \) times \( AX \) from the STAAD.Pro output = 12.97(11.70) = 151.7.

Table 554: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial capacity (kips)</td>
<td>154</td>
<td>151.7</td>
<td>1.5%</td>
</tr>
</tbody>
</table>

STAAD Input
The file \texttt{C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Column Compression Capacity 3.STD} is typically installed with the program.

### STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.

**START JOB INFORMATION**

- **ENGINEER DATE** 22-Sep-18

**END JOB INFORMATION**

* PAGE 3-28 OF 9TH ED. AISC-ASD. W12X40, 16 FT LONG.
* FA SHOULD BE APPROXIMATELY $154/11.80 = 13.05$ KSI

**UNIT FEET KIP**

**JOINT COORDINATES**

1 0 0 0; 2 0 16 0;

**MEMBER INCIDENCES**

1 1 2;

**MEMBER PROPERTY AMERICAN**

1 TABLE ST W12X40

**UNIT INCHES KIP**

**DEFINE MATERIAL START**

- **ISOTROPIC STEEL\_50\_KSI**
  - $E = 29000$
  - POISSON 0.3
  - DENSITY 0.000283
  - ALPHA 6.5e-06
  - DAMP 0.03

**TYPE STEEL**

- STRENGTH FY 50 FU 62 RY 1.5 RT 1.2

**END DEFINE MATERIAL**

**UNIT FEET KIP**

**CONSTANTS**

**MATERIAL STEEL\_50\_KSI ALL**

**SUPPORTS**

- 1 FIXED

**LOAD 1**

**JOINT LOAD**

- 2 FY -150

**PERFORM ANALYSIS**

**UNIT INCHES KIP**

**PARAMETER 1**

**CODE AISC**

**FYLD 36 ALL**

**TRACK 2 ALL**

**CHECK CODE ALL**

**FINISH**

---

### STAAD Output

**AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY -- PAGE NO. 3**

---

**STAAD.PRO CODE CHECKING - (AISC 9TH EDITION) v1.0**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>CODE</th>
<th>PROPERTY</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>AISC SECTIONS</td>
<td>ST W12X40</td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th></th>
<th>Y PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>IN INCH UNIT</td>
</tr>
<tr>
<td>1</td>
<td>AX = 11.70</td>
</tr>
<tr>
<td></td>
<td>--Z AV = 3.25</td>
</tr>
</tbody>
</table>
### V. AISC ASD - Square Tube Compression Capacity

Determine the axial compression capacity of a square 8x8x5/8 structural tube.

**Reference**


**Problem**

\[ F_y = 46 \text{ ksi} \]
\[ K_y L_y = K_z L_z = 25 \text{ft} \]

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output = 14.15(17.40) = 246.2.

Table 555: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial capacity (kips)</td>
<td>247</td>
<td>246.2</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Square Tube Compression Capacity.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.

START JOB INFORMATION
ENGINEER DATE 03-Sep-18
END JOB INFORMATION

* PAGE 3-41 OF 9TH ED. AISC-ASD. SQUARE TUBE 8X8X5/8, 25 FT LONG.
* FA SHOULD BE APPROX. 247/17.40 = 14.20 KSI

UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 25 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
TUBE
TUB808010
17.4 8 8 0.625 153 153 258 7.48 7.48
END
UNIT FEET KIP
UNIT INCHES KIP
MEMBER PROPERTY
1 UPTABLE 1 TUB808010
*1 TA ST TUB808010
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
Verification Examples
V.09 Steel Design

LOAD 1
JOINT LOAD
2 FY -240
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 46 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - (AISC 9TH EDITION) v1.0
*******************************************************************************

PARAMETER  IN KIP  INCH
L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 L0
L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 L0
STRESSES
|
--+-+---+---+---+---+---+---+---+---+---+---|
FA = 14.15
fa = 13.79
FCZ = 27.60
FTZ = 27.60
fby = 0.00
fey = 14.59
Fez = 14.59
FV = 18.40
fv = 0.00

MAX FORCE/ MOMENT SUMMARY (KIP-FEET)
-----------------------------
AXIAL  SHEAR-Y  SHEAR-Z  MOMENT-Y  MOMENT-Z
VALUE  240.0  0.0   0.0    0.0   0.0
LOCATION  0.0   0.0  0.0    0.0   0.0
LOADING  1   0     0   0     0

*******************************************************************************

* DESIGN SUMMARY (KIP-FEET) *

Verification Examples
V.09 Steel Design

STAAD.Pro
3997
User Manual
V. AISC ASD - Rectangular Tube Compression Capacity

Determine the axial compression capacity of a rectangular 7x5x3/8 structural tube.

Reference


Problem

\[ F_y = 46 \text{ ksi} \]

\[ K_y L_y = K_z L_z = 25 \text{ ft} \]

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output = 6.32(8.08) = 51.06.

Table 556: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial capacity (kips)</td>
<td>51</td>
<td>51.1</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Rectangular Tube Compression Capacity.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.

START JOB INFORMATION
ENGINEER DATE 03-Sep-18
END JOB INFORMATION
* PAGE 3-49 OF 9TH ED. AISC-ASD. RECTANGULAR TUBE 7X5X3/8, 25 FT LONG.
* FA SHOULD BE APPROX. 51/8.08 = 6.31 KSI
UNIT FEET KIP
JOINT COORDINATES
  1 0 0 0; 2 0 25 0;
MEMBER INCIDENCES
  1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
TUBE
### Verification Examples

**V.09 Steel Design**

TUB70506
8.08 7 5 0.375 52.2 30.8 64.2 3.97 2.98
END
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 TUB70506
*1 TA ST TUB70506
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
2 FY -50
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 46 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

---

**STAAD Output**

```
<table>
<thead>
<tr>
<th>Y PROPERTIES</th>
<th>IN INCH UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER 1 *</td>
<td>--------------</td>
</tr>
<tr>
<td>UPT</td>
<td>---------------</td>
</tr>
<tr>
<td>TUB70506</td>
<td>---------------</td>
</tr>
<tr>
<td>DESIGN CODE</td>
<td>--------------</td>
</tr>
<tr>
<td>AISC-1989</td>
<td>--------------</td>
</tr>
<tr>
<td>LENGTH (FT)= 25.00 ---&gt;</td>
<td></td>
</tr>
</tbody>
</table>

PARAMETER | L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 STRESSES |
IN KIP INCH | L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 IN KIP INCH |

KL/R-Y= 153.66 | L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 FA = 6.32 |
KL/R-Z= 118.03 | +L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 f0 = 6.19 |
UNL = 300.00  | L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 L0 FCZ = 27.60 |
```
**V. AISC ASD - Tee Compression Capacity**

Determine the axial compression capacity of a WT10.5x25.

**Reference**


**Problem**

\[ F_y = 36 \text{ ksi} \]
\[ K_y L_y = 5 \text{ ft} \]
\[ K_z L_z = 30 \text{ ft} \]

**Comparison**

The reported STAAD.Pro value is obtained by multiplying \( FA \) times \( AX \) from the STAAD.Pro output \( = 10.09(7.35) = 74.16 \).
### Table 557: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial capacity (kips)</td>
<td>74</td>
<td>74.16</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

#### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Tee Compression Capacity.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.

START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* PAGE 3-93 OF 9TH ED. AISC-ASD. TEE SHAPE WT10.5X25, KL Z-AXIS = 30 FT
* KL Y-AXIS = 5 FT, FA SHOULD BE APPROX. 74/7.35 = 10.07 KSI
UNIT FEET KIP
JOIN COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE T W21X50
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOIN LOAD
2 FY -70
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
LZ 360 ALL
LY 60 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
### STAAD Output

**STAAD.PRO CODE CHECKING - (AISC 9TH EDITION) v1.0**

```
----------------------------------------
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>AISC SECTIONS</th>
<th>DESIGN CODE</th>
<th>AISC-1989</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>T W21X50</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PROPERTIES</th>
<th>IN INCH UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>AX</td>
<td>7.35</td>
</tr>
<tr>
<td>AY</td>
<td>2.83</td>
</tr>
<tr>
<td>AZ</td>
<td>2.63</td>
</tr>
<tr>
<td>SY</td>
<td>3.81</td>
</tr>
<tr>
<td>SZ</td>
<td>10.63</td>
</tr>
<tr>
<td>RY</td>
<td>1.30</td>
</tr>
<tr>
<td>RZ</td>
<td>3.29</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LENGTH (FT)</th>
<th>10.00</th>
</tr>
</thead>
</table>

**PARAMETER**

<table>
<thead>
<tr>
<th>IN KIP INCH</th>
<th>IN KIP INCH</th>
<th>STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td>L0 L0 L0 L0</td>
<td>L0 L0 L0 L0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>KL/R-Y</th>
<th>85.76</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Z</td>
<td>109.54</td>
</tr>
<tr>
<td>UNL</td>
<td>120.00</td>
</tr>
<tr>
<td>CB</td>
<td>1.00</td>
</tr>
<tr>
<td>CMY</td>
<td>0.85</td>
</tr>
<tr>
<td>CMZ</td>
<td>0.85</td>
</tr>
<tr>
<td>FYLD</td>
<td>36.00</td>
</tr>
<tr>
<td>NSF</td>
<td>1.00</td>
</tr>
<tr>
<td>DFF</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**MAX FORCE/ MOMENT SUMMARY (KIP-FEET)**

<table>
<thead>
<tr>
<th>VALUE</th>
<th>70.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOADING</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**DESIGN SUMMARY (KIP-FEET)**

<table>
<thead>
<tr>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>PASS</td>
<td>AISC- H1-1</td>
<td>0.944</td>
<td>1</td>
</tr>
<tr>
<td>70.00 C</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>
Determine the bending (transverse load carrying) capacity of a W10X45 shape.

Reference


**Problem**

\[ F_y = 36 \text{ ksi} \]

\[ L_{u} = 6 \text{ ft} \]

**Comparison**

The reported STAAD.Pro value is obtained by dividing the applied load (using the table capacity) by the reported STAAD.Pro critical ratio \( \frac{17}{1.002} = 16.97 \).

**Table 558: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable uniform load (kips/ft)</td>
<td>17</td>
<td>16.97</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

 STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-34 OF
 START JOB INFORMATION
 ENGINEER DATE 22-Sep-18
 END JOB INFORMATION
 * 9TH ED. AISC-ASD EXAMPLE 5. GOVERNING CONDITION IS SHEAR.
 * RATIO SHOULD BE APPROXIMATELY 1.0
 UNIT FEET KIP
 JOINT COORDINATES
 1 0 0 0; 2 6 0 0;
 MEMBER INCIDENCES
 1 1 2;
 MEMBER PROPERTY AMERICAN
 1 TABLE ST W10X45
 UNIT INCHES KIP
 DEFINE MATERIAL START
 ISOTROPIC MATERIAL1
 E 29000
 POISSON 0.3
 END DEFINE MATERIAL
 UNIT FEET KIP
 CONSTANTS
 MATERIAL MATERIAL1 ALL
 SUPPORTS
 1 PINNED
 2 FIXED BUT MZ
 LOAD 1
 MEMBER LOAD
 1 UNI GY -17
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
TRACK 2 ALL
SHEAR 1 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>LOCATION</th>
<th>LOADING</th>
<th>VALUE</th>
<th>61.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>76.5 (KIP-FEET)</td>
<td>0.0</td>
<td>0.0</td>
<td>51.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td></td>
</tr>
</tbody>
</table>

MAX FORCE/MOMENT SUMMARY (KIP-FEET)

| VALUE | 61.0 | 51.0 | 0.0 | 0.0 | 0.0 | 76.5 |
| AXIAL | 0.0 | 0.0 | 0.0 | 0.0 | 0.0 | 3.0 |
| LOCATION | 0.0 | 0.0 | 0.0 | 0.0 | 0.0 | 1.0 |

DESIGN SUMMARY (KIP-FEET)
V. AISC ASD - Wide Flange Beam Load Capacity 1

Determine the bending (transverse load carrying) capacity of a W16x40 shape.

Reference


Problem

\[ F_y = 36 \text{ ksi} \]

\[ L_{u} = 6\text{ft} \]

Comparison

The reported STAAD.Pro value is obtained by multiplying the FCZ and SZ values from the STAAD.Pro output:

\[ 23.76(64.71) = 1,537.5 \text{ in·kip} = 128.1. \]

Table 559: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending capacity (ft-kips)</td>
<td>128</td>
<td>128.1</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Wide Flange Beam Load Capacity 1.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-5 OF START JOB INFORMATION

ENGINEER DATE 22-Sep-18

END JOB INFORMATION

* 9TH ED. AISC-ASD CODE EXAMPLE 1. RATIO SHOULD BE A LITTLE UNDER 1.0.
* CAPACITY SHOULD BE 128 KIP-FT. (SZ*FCZ SHOULD BE 128 KIP-FT).

UNIT FEET KIP

JOINT COORDINATES
1 0 0 0; 2 10 0 0;

MEMBER INCIDENCES
1 1 2;

MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40

UNIT INCHES KIP

DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
FYLD 36 ALL
UNL 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>Y</th>
<th>PROPERTIES IN INCH UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER 1</td>
<td>AISC SECTIONS</td>
</tr>
<tr>
<td></td>
<td>ST W16X40</td>
</tr>
<tr>
<td>DESIGN CODE</td>
<td></td>
</tr>
<tr>
<td>AISC-1989</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>125.0 (KIP-FEET)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>L1</th>
<th>L1</th>
<th>L1</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>76.68</td>
<td>L1</td>
<td></td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>18.11</td>
<td></td>
<td></td>
</tr>
<tr>
<td>UNL</td>
<td>72.00</td>
<td></td>
<td>L1</td>
</tr>
<tr>
<td>CB</td>
<td>1.00</td>
<td>L1</td>
<td></td>
</tr>
<tr>
<td>CMY</td>
<td>0.85</td>
<td>L1</td>
<td></td>
</tr>
<tr>
<td>CMZ</td>
<td>0.85</td>
<td></td>
<td>L1</td>
</tr>
<tr>
<td>FYLD</td>
<td>36.00</td>
<td></td>
<td>L0</td>
</tr>
<tr>
<td>NSF</td>
<td>1.00</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>DFF</td>
<td>0.00</td>
<td>-6.9</td>
<td></td>
</tr>
</tbody>
</table>

MAX FORCE/ MOMENT SUMMARY (KIP-FEET)

<table>
<thead>
<tr>
<th>(KL/R)max = 76.68</th>
<th>(WITH LOAD NO.)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
V. AISC ASD - Wide Flange Beam Load Capacity 2

Determine the bending (transverse load carrying) capacity of a W16x40 shape.

Reference

Problem

\[ F_y = 36 \text{ ksi} \]

\[ L_{uc} = 9 \text{ ft (Unbraced length of compression flange)} \]

Comparison

The reported STAAD.Pro value is obtained by multiplying the FCZ and SZ values from the STAAD.Pro output:

\[ 21.60 \times 64.71 = 1397.7 \text{ in·kip} = 116.5 \text{ ft·kips} \]

Table 560: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending capacity</td>
<td>116.5</td>
<td>116.5</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Wide Flange Beam Load Capacity 2.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-6 OF START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD CODE EXAMPLE 2.
* CAPACITY SHOULD BE 116.5 KIP-FT. (SZ*FCZ SHOULD BE 116.5 KIP-FT).
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 9 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.94
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>IN KIP  INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>69.01</td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>16.30</td>
</tr>
<tr>
<td>UNL</td>
<td>108.00</td>
</tr>
<tr>
<td>CB</td>
<td>1.00</td>
</tr>
</tbody>
</table>

116.5 (KIP-FEET)

<table>
<thead>
<tr>
<th>Y PROPERTIES</th>
<th>IN INCH UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>AX</td>
<td>11.80</td>
</tr>
<tr>
<td>AZ</td>
<td>4.73</td>
</tr>
<tr>
<td>SY</td>
<td>8.26</td>
</tr>
<tr>
<td>SZ</td>
<td>64.75</td>
</tr>
<tr>
<td>RY</td>
<td>1.56</td>
</tr>
<tr>
<td>RZ</td>
<td>6.63</td>
</tr>
</tbody>
</table>
V. AISC ASD - MC Beam Load Capacity

Determine the bending (transverse load carrying) capacity of an MC18x58 channel. The beam is laterally supported along its entire length.

Reference


Problem

\[ F_y = 36 \text{ ksi} \]
\[ L = 25 \text{ ft} \]

Comparison

The reported STAAD.Pro value is obtained by dividing the applied load (using the table capacity) by the reported STAAD.Pro critical ratio \(=\frac{43}{0.995} = 43.2\).

**Table 561: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total allowable uniform load (kips/ft)</td>
<td>43</td>
<td>43.2</td>
<td>0.5%</td>
</tr>
</tbody>
</table>
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - MC Beam Load Capacity.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-80 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD. LATERALLY SUPPORTED MC18X58 CHANNEL, 25FT LONG
* LOAD CAPACITY = 43 KIPS. RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST MC18X58
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
* LOAD INTENSITY = 43/25 = 1.72 KIP/FT
1 UNI GY -1.72
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
UNL 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>Mem</th>
<th>Section Type</th>
<th>AX</th>
<th>AY</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ST MC18X58</td>
<td>17.10</td>
<td>9.98</td>
</tr>
</tbody>
</table>
### V. AISC ASD - Wide Flange Beam Load Capacity 3

Determine the bending (transverse load carrying) capacity of a W16x45 shape.

**Reference**


**Problem**

\[ F_y = 36 \text{ ksi} \]
L = 20ft
L_{uc} = 5ft (Unbraced length of compression flange)

Comparison
The reported STAAD.Pro value is obtained by dividing the applied load (using the table capacity) by the reported
STAAD.Pro critical ratio = $\frac{58}{1.008} = 57.54$.

Table 562: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total allowable uniform load (kips)</td>
<td>58</td>
<td>57.54</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Wide Flange Beam Load Capacity 3.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-34 OF START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD EXAMPLE 4.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
  1 0 0 0; 2 20 0 0;
MEMBER INCIDENCES
  1 1 2;
MEMBER PROPERTY AMERICAN
  1 TABLE ST W16×45
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
  E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
  1 PINNED
  2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
  1 UNI GY -2.9
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
TRACK 2 ALL
### Verification Examples

#### V.09 Steel Design

**STAAD Output**

```
STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)  v1.0
********************************************

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>DESIGN CODE</th>
<th>PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST  W16X45</td>
<td>AISC-1989</td>
<td></td>
</tr>
</tbody>
</table>

**PARAMETER**

<table>
<thead>
<tr>
<th>IN KIP</th>
<th>IN INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>152.83</td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>36.16</td>
</tr>
<tr>
<td>UNL</td>
<td>60.00</td>
</tr>
<tr>
<td>CB</td>
<td>1.00</td>
</tr>
<tr>
<td>CMY</td>
<td>0.85</td>
</tr>
<tr>
<td>CMZ</td>
<td>0.85</td>
</tr>
<tr>
<td>FYLD</td>
<td>36.00</td>
</tr>
<tr>
<td>NSF</td>
<td>1.00</td>
</tr>
<tr>
<td>DFF</td>
<td>0.00</td>
</tr>
<tr>
<td>dff</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**145.0 (KIP-FEET)**

<table>
<thead>
<tr>
<th>L1</th>
<th>L1</th>
<th>L1</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**MAX FORCE/ MOMENT SUMMARY (KIP-FEET)**

<table>
<thead>
<tr>
<th>VALUE</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**CRITICAL COND/ RATIO/ LOADING**

<table>
<thead>
<tr>
<th>FX</th>
<th>MY</th>
<th>MZ</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00 T</td>
<td>0.00</td>
<td>-145.00</td>
<td>10.00</td>
</tr>
</tbody>
</table>
```
V. AISC ASD - Select Wide Flange Beam 1

Determine the optimum W shape that spans 30 ft, and is braced at 7.5 ft intervals.

Reference


Problem

Three concentrated loads of 20 kips each at quarter span points. Desired beam depth of 18 in.

\[ F_y = 36 \text{ ksi} \]

Comparison

**Table 563: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selected section</td>
<td>W18x86</td>
<td>W18x86</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Select Wide Flange Beam 1.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-35 OF

START JOB INFORMATION

ENGINEER DATE 22-Sep-18

END JOB INFORMATION

* 9TH ED. AISC-ASD EXAMPLE 6.
* RATIO SHOULD BE APPROXIMATELY 1.0

UNIT FEET KIP

JOINT COORDINATES
1 0 0 0; 2 30 0 0;

MEMBER INCIDENCES
1 1 2;

MEMBER PROPERTY AMERICAN
1 TABLE ST W18X35

UNIT INCHES KIP

DEFINE MATERIAL START

ISOTROPIC MATERIAL1

E 29000

POISSON 0.3

END DEFINE MATERIAL

UNIT FEET KIP

CONSTANTS

MATERIAL MATERIAL1 ALL

SUPPORTS
1 PINNED
2 FIXED BUT MZ

LOAD 1
MEMBER LOAD
1 CON GY -20 7.5
1 CON GY -20 15
1 CON GY -20 22.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
DMAX 19 ALL
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
UNL 90 ALL
TRACK 2 ALL
SELECT OPTIMIZED
FINISH

STAAD Output

STAAD.PRO MEMBER SELECTION - (AISC 9TH EDITION) v1.0
***************************************************************************************
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Y        PROPERTIES</td>
</tr>
<tr>
<td>*************</td>
</tr>
<tr>
<td>*</td>
</tr>
<tr>
<td>MEMBER   1  *</td>
</tr>
<tr>
<td>*</td>
</tr>
<tr>
<td>DESIGN CODE *</td>
</tr>
<tr>
<td>AISC-1989  *  ===============================   ===</td>
</tr>
<tr>
<td>*                                                SZ = 166.30</td>
</tr>
<tr>
<td>*</td>
</tr>
<tr>
<td>*************                                                RZ =   7.78</td>
</tr>
</tbody>
</table>

|                300.0 (KIP-FEET)                                          |
|PARAMETER        |                   L1                      STRESSES     |
|IN KIP  INCH     |               L1      L1              IN KIP  INCH |
|---------------  +           L1              L1              -------------|
| KL/R-Y= 136.88  |                                           FA  =   7.97 |
| KL/R-Z=  46.29  +                               L1          fa  =   0.00 |
| UNL   =  90.00  |       L1                                  FCZ =  23.76 |
| CB    =   1.00  +                                           FTZ =  23.76 |
| CMY   =   0.85  |   L1                              L1      FCY =  27.00 |
| CMZ   =   0.85  +                                           FTY =  27.00 |
| FYLD  =  36.00  |L0                                     L0  fbz =  21.65 |
| NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
| DFF   =   0.00 -16.7                                        Fey =   7.97 |
| dff=      0.00               ABSOLUTE MZ ENVELOPE           Fez =  69.68 |
| (KL/R)max = 136.88              (WITH LOAD NO.)             FV  =  14.40 |

MAX FORCE/ MOMENT SUMMARY (KIP-FEET)

<table>
<thead>
<tr>
<th></th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOMENT-Y</th>
<th>MOMENT-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>VALUE</td>
<td>0.0</td>
<td>30.0</td>
<td>0.0</td>
<td>0.0</td>
<td>300.0</td>
</tr>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>15.0</td>
</tr>
</tbody>
</table>
V. AISC ASD - Select Wide Flange Beam 2

Determine a 14in deep section to carry a load of 1 kip/ft over a span of 25 ft.

Reference


Problem

The beam is laterally supported throughout its length.

\[ F_y = 36 \text{ ksi} \]

Comparison

Table 564: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selected section</td>
<td>W14x30</td>
<td>W14x30</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Select Wide Flange Beam 2.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-35 OF START JOB INFORMATION

ENGINEER DATE 22-Sep-18

END JOB INFORMATION

* 9TH ED. AISC-ASD EXAMPLE 7. LATERALLY SUPPORTED BEAM.

UNIT FEET KIP

JOINT COORDINATES

1 0 0 0; 2 25 0 0;

MEMBER INCIDENCES

1 1 2;

MEMBER PROPERTY AMERICAN

* TRIAL SECTION SIZE:
### DEFINE MATERIAL

**ISOTROPIC MATERIAL1**

- $E = 29000$
- $\nu = 0.3$

### SUPPORTS

- 1 PINNED
- 2 FIXED BUT MZ

### LOAD 1

**MEMBER LOAD**

- 1 UNI GY -1

### ANALYSIS

**PARAMETER 1**

- CODE AISC
- $D_{MAX} = 15$ ALL
- MAIN 1 ALL
- BEAM 1 ALL
- FYLD 36 ALL
- UNL 1 ALL
- TRACK 2 ALL

*CHECK CODE ALL*

**SELECT OPTIMIZED**

**FINISH**

### STAAD Output

<table>
<thead>
<tr>
<th>PARAMETERS</th>
<th>KIP-FT</th>
<th>STRESSES</th>
<th>KIP-FT</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{KL}{R-Y} = 201.59$</td>
<td>$L_1$</td>
<td>$L_1$</td>
<td>$FA = 3.67$</td>
</tr>
<tr>
<td>$\frac{KL}{R-Z} = 52.32$</td>
<td>$L_1$</td>
<td>$L_1$</td>
<td>$Fa = 0.00$</td>
</tr>
<tr>
<td>UNL = 1.00</td>
<td>$L_1$</td>
<td>$FTZ = 23.76$</td>
<td></td>
</tr>
<tr>
<td>CB = 1.00</td>
<td>$L_1$</td>
<td>$FTZ = 23.76$</td>
<td></td>
</tr>
<tr>
<td>CMY = 0.85</td>
<td>$L_1$</td>
<td>$FCY = 27.00$</td>
<td></td>
</tr>
<tr>
<td>CMZ = 0.85</td>
<td>$L_0$</td>
<td>$FTY = 27.00$</td>
<td></td>
</tr>
<tr>
<td>FYLD = 36.00</td>
<td></td>
<td>$fbz = 22.23$</td>
<td></td>
</tr>
</tbody>
</table>
V. AISC ASD - Select Wide Flange Beam 3

Determine an optimum section to carry a moment of 220 ft·kips plus its selfweight and spanning 35 ft between supports.

Reference


Problem

\[ F_y = 36 \text{ ksi} \]

\[ C_b = 1.0 \]

Maximum unbraced length of compression flange is 15 ft.

Comparison

Table 565: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selected section</td>
<td>W24x68</td>
<td>W24x68</td>
<td>none</td>
</tr>
</tbody>
</table>
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Select Wide Flange Beam 3.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-147
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* OF 9TH ED. AISC-ASD. EXAMPLE 10.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
*TRIAL SECTION
1 TABLE ST W18X35
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
SELFWEIGHT Y -1
MEMBER LOAD
1 CON GY -22 10
1 CON GY -22 25
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
UNL 180 ALL
TRACK 2 ALL
*CHECK CODE ALL
SELECT OPTIMIZED
FINISH

STAAD Output

*** STAAD.PRO MEMBER SELECTION - (AISC 9TH EDITION) ***
### Member Properties

<table>
<thead>
<tr>
<th>MEMBER 1</th>
<th>AISC SECTIONS</th>
<th>ST W24X68</th>
</tr>
</thead>
<tbody>
<tr>
<td>DESIGN CODE</td>
<td>AISC-1989</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>PROPERTIES</strong></td>
<td><strong>IN INCH UNIT</strong></td>
<td></td>
</tr>
<tr>
<td>AX = 20.10</td>
<td>AZ = 7.81</td>
<td></td>
</tr>
<tr>
<td>--Z</td>
<td>SY = 15.70</td>
<td></td>
</tr>
<tr>
<td></td>
<td>AZ = 7.01</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SY = 15.43</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LENGTH (FT) = 35.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Parameter

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>L1</th>
<th>L1</th>
<th>L1</th>
</tr>
</thead>
<tbody>
<tr>
<td>IN KIP INCH</td>
<td>-----</td>
<td>-----</td>
<td>----</td>
</tr>
<tr>
<td>KL/R-Y= 224.42</td>
<td>FA  = 2.97</td>
<td></td>
<td></td>
</tr>
<tr>
<td>KL/R-Z= 44.02</td>
<td>fa  = 0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>UNL  = 180.00</td>
<td>FCZ = 18.70</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CB   = 1.00</td>
<td>FCY = 27.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CMY  = 0.85</td>
<td>FYT = 27.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CMZ  = 0.85</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FYLD = 36.00</td>
<td>L0</td>
<td>fby = 0.00</td>
<td></td>
</tr>
<tr>
<td>NSF  = 1.00</td>
<td>Fez = 77.07</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DFF  = 0.00</td>
<td>(WITH LOAD NO.)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>dff= 0.00</td>
<td>Fey = 2.97</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(KL/R)max = 224.42</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Stresses

<table>
<thead>
<tr>
<th></th>
<th>L1</th>
<th>L1</th>
</tr>
</thead>
<tbody>
<tr>
<td>IN KIP INCH</td>
<td>-----</td>
<td>----</td>
</tr>
<tr>
<td>AXIAL</td>
<td>SHEAR-Y</td>
<td>SHEAR-Z</td>
</tr>
<tr>
<td>VALUE</td>
<td>0.0</td>
<td>23.5</td>
</tr>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOADING</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

### Design Summary (KIP-Feet)

<table>
<thead>
<tr>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>PASS</td>
<td>AISC- H1-3</td>
<td>0.967</td>
<td>1</td>
</tr>
<tr>
<td>0.00 T</td>
<td>0.00</td>
<td>-232.69</td>
<td>17.50</td>
</tr>
</tbody>
</table>

---

AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-1 -- PAGE NO. 4

---

ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)

---

STAAD.Pro 4020 User Manual
V. AISC ASD - Compression and Biaxial Bending

Determine the suitability of a W14x109 shape to resist axial compression and biaxial bending.

Reference


Problem

\[
P = 200 \text{ kips}
\]
\[
M_z = 120 \text{ ft·kips (strong axis)}
\]
\[
M_y = 40 \text{ ft·kips (weak axis)}
\]
\[
KL = 14 \text{ ft}
\]
\[
C_m = 0.85
\]

Maximum unbraced length of compression flange is 15 ft.

Comparison

Table 566: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selected section</td>
<td>OK</td>
<td>OK (stress ratio = 0.93)</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - Compression and Biaxial Bending.STD is typically installed with the program.

STAAD SPACE AISC VERIFICATION PROBLEM FOR COMBINED AXIAL + BENDING.

* PAGE 3-11 OF 9TH ED. AISC-ASD. EXAMPLE 4.
* RATIO SHOULD BE APPROXIMATELY 1.0

UNIT FEET KIP

JOINCOORDINATES
1 0 0 0; 2 0 14 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X109
UNIT INCHES KIP

DEFINE MATERIAL START

ISCOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -200
2 MZ 120
2 MX 40
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISC
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>Y PROPERTIES</th>
<th>IN INCH UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER 1</td>
<td>------------</td>
</tr>
<tr>
<td>AISC SECTIONS</td>
<td></td>
</tr>
<tr>
<td>ST W14X109</td>
<td></td>
</tr>
<tr>
<td>AX = 32.00</td>
<td></td>
</tr>
<tr>
<td>AZ = 16.00</td>
<td></td>
</tr>
<tr>
<td>SY = 61.23</td>
<td></td>
</tr>
<tr>
<td>SZ = 173.43</td>
<td></td>
</tr>
<tr>
<td>RY = 3.74</td>
<td></td>
</tr>
<tr>
<td>RZ = 6.22</td>
<td></td>
</tr>
<tr>
<td>KL/R-Y= 44.95</td>
<td></td>
</tr>
<tr>
<td>KL/R-Z= 26.99</td>
<td></td>
</tr>
<tr>
<td>UNL = 168.00</td>
<td></td>
</tr>
<tr>
<td>CB = 1.00</td>
<td></td>
</tr>
<tr>
<td>CMY = 0.85</td>
<td></td>
</tr>
<tr>
<td>CMZ = 0.85</td>
<td></td>
</tr>
<tr>
<td>FYLD = 36.00</td>
<td></td>
</tr>
<tr>
<td>L1 L1 L1 FA  = 18.78</td>
<td></td>
</tr>
<tr>
<td>L1 L1 L1 fa  = 6.25</td>
<td></td>
</tr>
<tr>
<td>L1 L1 L1 FCZ = 23.76</td>
<td></td>
</tr>
<tr>
<td>L1 L1 L1 FTY = 27.00</td>
<td></td>
</tr>
<tr>
<td>L1 L1 L1 fbz = 8.30</td>
<td></td>
</tr>
</tbody>
</table>
V. AISC ASD - Angle in Compression

Determine the allowable compressive load on a single L2x2x1/4 loaded by a gusset plate attached to one leg with eccentricities from the centroid along both axes.

Reference


Problem

KL = 40 in

Comparison

Table 567: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable compressive load (kips)</td>
<td>4.5</td>
<td>Using a P of 4.5 kip yields a ratio of 0.898</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION + BIAXIAL BENDING
* PAGE 3-55 OF 9TH ED. AISC-ASD. SINGLE ANGLE 2X2X1/4
* UNBRACED LENGTH IS 40 INCHES.
UNIT KIP FEET
JOINT COORD
1 0 0 0 ; 2 0 5 0
MEMB INCI
1 1 2
UNIT INCH
MEMB PROP AMERICAN
1 TA ST L20204
CONST
E STEEL ALL
POISS STEEL ALL
SUPP
1 FIXED
LOAD 1
JOINT LOAD
2 FY -4.5
* MZ = P * 0.277 = 1.2465 KIP-INCH
* MX = P * 0.854 = 3.843 KIP-INCH
2 MZ 1.2465
2 MX -3.843
PERF ANALY
UNIT INCH
PARAM
CODE AISC
FYLD 50 ALL
UNT 40 ALL
UNB 40 ALL
LY 40 ALL
LZ 40 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

PEND NO. |
--- | --- |
1 | 1. STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION + BIAXIAL BENDING |
**INPUT FILE: AISC ASD - Angle in Compression.STD**

2. * PAGE 3-55 OF 9TH ED. AISC-ASD. SINGLE ANGLE 2X2X1/4
3. * UNBRACED LENGTH IS 40 INCHES.
4. UNIT KIP FEET
5. JOINT COORD
6. 1 0 0 0 ; 2 0 5 0
7. MEMB INCI
8. 1 1 2
9. UNIT INCH
10. MEMB PROP AMERICAN
11. 1 TA ST L20204
12. CONST
13. E STEEL ALL
14. POISS STEEL ALL
15. SUPP
16. 1 FIXED
17. LOAD 1
18. JOINT LOAD
19. 2 FY -4.5
20. * MZ = P * 0.277 = 1.2465 KIP-INCH
21. * MX = P * 0.854 = 3.843 KIP-INCH
22. 2 MZ 1.2465
23. 2 MX -3.843
24. PERF ANALY

**P R O B L E M   S T A T I S T I C S**

<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>NUMBER OF JOINTS  2</td>
</tr>
<tr>
<td>NUMBER OF PLATES  0</td>
</tr>
<tr>
<td>NUMBER OF SURFACES 0</td>
</tr>
</tbody>
</table>

Using 64-bit analysis engine.

**AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION + BIAXIA -- PAGE NO.**

2

* PAGE 3-55 OF 9TH ED. AISC-ASD. SINGLE ANGLE

SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 6
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

25. UNIT INCH
26. PARAM
27. CODE AISC
28. FYLD 50 ALL
29. UNT 40 ALL
30. UNB 40 ALL
31. LY 40 ALL
32. LZ 40 ALL
33. TRACK 2 ALL
34. CHECK CODE ALL

**STEEL DESIGN**

AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION + BIAXIA -- PAGE NO.

3

* PAGE 3-55 OF 9TH ED. AISC-ASD. SINGLE ANGLE

STAAD.Pro CODE CHECKING - (AISC 9TH EDITION) v1.0

***---------------------------------------------------------------------------------------------------

<table>
<thead>
<tr>
<th>Y</th>
<th>PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>INCH UNIT</td>
<td></td>
</tr>
<tr>
<td>MEMBER 1 *</td>
<td>AISC SECTIONS</td>
</tr>
<tr>
<td>*</td>
<td>ST L20204</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>=</th>
<th>==</th>
<th>==</th>
</tr>
</thead>
<tbody>
<tr>
<td>AX = 0.94</td>
<td></td>
<td></td>
</tr>
<tr>
<td>--Z AY = 0.33</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
DESIGN CODE * | | == | ==  
AISC-1989 * =============== == | ==  
* | ==<---LENGTH (FT)= 5.00 -->> |  
***************  
0.1 (KIP-FEET)  
PARAMETER | L1 L1 L1 STRESSES  
IN KIP INCH | L1 L1 L1 IN KIP INCH  
---------- +  
L1 L1 L1 ----------  
KL/R-Y= 52.22 | L1 L1 L1 FA = 13.92  
KL/R-Z= 103.36 + L1 L1 L1 fa = 4.77  
UNL = 40.00 | L1 L1 L1 L1 L1 L1 L1 L1 FCZ = 33.00  
CB = 1.00 +L1 L1 L1 L1 L1 L1 L1 L1 FTZ = 33.00  
CMY = 0.85 | L1 L1 L1 L1 L1 L1 L1 L1 FCY = 33.00  
CMZ = 0.85 +L1 L1 L1 L1 L1 L1 L1 L1 FTY = 33.00  
FYLD = 50.00 | L1 L1 L1 L1 L1 L1 L1 L1 fbz = 7.11  
NSF = 1.00 +--------------------------| fby = 9.89  
DFF = 0.00 0.1  
dff= 0.00 ABSOLUTE MZ ENVELOPE  
(KL/R)max = 103.36 (WITH LOAD NO.)  
FV = 20.00  
fv = 0.00  

MAX FORCE/ MOMENT SUMMARY (KIP-FEET)  
-------------------------  
VALUE 4.5 0.0 0.0 0.3 0.1  
LOCATION 0.0 0.0 0.0 0.0 0.0  
LOADING 1 0 0 1 1  

*******************************  
  
*********** END OF THE STAAD.Pro RUN ***********  
Verification Examples  
V.09 Steel Design  
STAAD.Pro  
4026  
User Manual
V. AISC ASD - 2D Frame Validation

Determine the allowable stresses (per 1989 AISC code) for the members of the structure as shown in the figure. Also, perform a code check for these members based on the results of the analysis.

Reference


Problem

Members 1, 2 = W12X26, Members 3, 4 = W14X43
Members 5, 6, 7 = W16X36, Memb 8= L40404, Memb 9 = L50506

F = 15 kip (wind load)
w = 2 kip/ft (dead load + live load)
Steel: Fy = 36ksi

Hand Calculation


Member 1
Size W12x26, L = 10 ft., a = 7.65 in², Sz = 33.39 in³
Per Section F2 is the lower value for the yielding and lateral-torsional buckling limit states:
Mₙ = Mₚ = FᵧZₓ = 36ksi × 37.2 in³ = 1,339 in·k = 111.3 ft·k

\[
L_p = 1.76r_y\sqrt{\frac{E}{F_y}} = 1.76\left(1.51\text{in}\right)\sqrt{\frac{29,000\text{ksi}}{36\text{ksi}}} = 75.4\text{in}
\]

From observation Load case 1 will govern,
Fₓ = 25.0 kip (compression), Mₓ = 56.5 ft·kip
Area of compressive flange = 6.49*0.38 = 2.466 in²
Allowable bending stress, per Clause F1-8 (page 5-47 of Manual)
\[
f_b = 12 \cdot 1000 \cdot 1.0/[10 \cdot 12 \cdot (12.22/2.466)] = 20.1817 \text{ksi}
\]
(\(k/l/r\)ᵧ = 120/1.5038 = 79.8, so \(f_a = 15.38 \text{ksi}\)

Stresses per Table C-36 (page 3-16 of manual)
\[
f_a = F_x/7.65 = 3.268
\]
\[
f_b = 56.5 \cdot 12/33.39 = 20.31 \text{ksi}
\]
(\(k/l/r\)z = 120/5.1639 = 23.238, so \(F'_{ez} = 276.54 \text{ksi}\)

Equation H1-1, page 5-54 of Manual
\[
3.268/15.38 + 0.85 \cdot 20.31 / [(1-3.268/276.54) \cdot 20.1817] = 1.078
\]
Equation H1-2, page 5-54 of Manual.
\[
3.268/(0.6 \cdot 36) + 20.31/20.1817 = 1.1576
\]
Therefore, equation H1-2 governs and ratio = 1.1576

Member 2
Size W12x26, L = 5 ft., a = 7.65 in², Sz = 33.39 in³
From observation load case 1 will govern,
Fₓ = 8.71 kip (compression), Mₓ = 56.50 ft·kip
Since L is less than Lₜ = 6.85ft, per Clauses F1-1 & F1-2 (page 5-45 of Manual)
\[
f_b = 0.66 \cdot 36 = 23.76 \text{ksi}
\]
(\(k/l/r\)ᵧ = 60/1.5038 = 39.90, so \(f_a = 19.19 \text{ksi}\)

Per Table C-36 (page 3-16 of Manual)
\[
f_a = 8.71/7.65 = 1.1385, \ f_b = 56.5 \cdot 12/33.39 = 20.31 \text{ksi}
\]
Since \(f_a/F_a\) less than 0.15, apply equation H1-3 (page 5-54 of Manual)
\[
1.1385/19.19 + 20.31/23.76 = .0593 + .8548 = 0.9141
\]

Member 3
Size W14X43, L = 11 ft, a = 12.6 in², Sz = 62.7 in³
From observation load case 3 will govern,
\[ F_x = 25.5 \text{ kip (compression), } M_z = 112.173 \text{ ft-kip} \]
Referring to clause F1-2, page 5-45 of Manual.
\[ L_c = 8.4 \text{ ft. Therefore} \]
\[ f_b = 0.6 \cdot 36 = 21.6 \text{ ksi} \]
\[ (kI/r)y = 132/1.8941 = 69.69, \]
so \[ f_a = 16.46 \text{ ksi per Table C-36 (page 3-16 of Manual)} \]
\[ f_a = 25.5/12.6 = 2.024 \]
\[ f_b = 112.173 \cdot 12/62.66 = 21.48 \text{ ksi} \]
since \[ f_a/F_a \text{ less than 0.15, use equation H1-3, page 5-54 of Manual} \]
\[ 2.024/16.46 + 21.48/21.6 = 0.123 + 0.994 = 1.117 \]
Member 4
Size W14x43, L = 4 ft, a = 12.6 in², Sz = 62.7 in³
From observation, load case 3 will govern,
\[ F_x = 8.75 \text{ kip (tension), } M_z = 112.173 \text{ ft-kip} \]
Since L is less than \( L_c = 8.4 \text{ ft} \)
\[ f_b = 0.66 \cdot 36 = 23.76 \text{ ksi} \]
Clause F1-1 (page 5-45 of Manual)
\[ f_a = 8.75/12.6 = 0.694 \]
\[ f_b = 112.73x12/62.66 = 21.48 \text{ ksi} \]
Combined tension and bending, use equation H2-1, page 5-55 of Manual.
\[ 0.694/(0.6 \cdot 36) + 21.48/23.76 = 0.032 + 0.904 = 0.936 \]
Member 5
Size W16x36, L = 5 ft, a = 10.6 in², Sz = 56.49 in³
From observation, load case 3 will govern.
\[ F_x = 14.02 \text{ kip (compression), } M_z = 57.04 \text{ ft-kip} \]
Since L is less than \( L_c = 7.37 \text{ (per Clause F1-2, page 5-45 of Manual)} \)
\[ f_b = 0.66 \cdot 36 = 23.76 \text{ ksi} \]
\[ (kI/r)y = 60./1.52 = 39.47 \]
so \[ f_a = 19.23 \text{ ksi per Table C-36 (page 3-16 of Manual)} \]
\[ f_a = 14.02/10.6 = 1.32 \]
\[ f_b = 57.04 \cdot 12/56.5 = 12.12 \text{ ksi} \]
Since \[ f_a/F_a \text{ less than 0.15, use equation H1-3, page 5-54 of Manual} \]
1.32/19.23 + 12.12/23.76 = 0.069 + 0.510 = 0.579

Member 6
Size W16x36, L = 16 ft, a = 10.6 in², Sz = 56.49 in³
From observation, load case 1 will govern. Forces at midspan are
Fₓ = 5.65 kip (compression), Mₑ = 71.25 ft·kip
From Chapter F of the AISC ASD 9th ed. specs., with Cb = 1.0,
(102,000Cb/Fy)₁/² = 53.229
(510,000Cb/Fy)₁/² = 119.02
L/rT = 192/1.79 = 107.26
53.229 < 107.26 < 119.02
Therefore Fₑ (as per F1-6, page 5-47 of Manual)
[(2/3) – 36 · 107.26 · 107.26/(1,530,000)] · 36 = 14.25 ksi
(K/I/r)y = 192/1.5203 = 126.29
so fₓ = 9.36 per Table C-36 (page 3-16 of Manual)

fₑ = 5.65/10.6 = 0.533
fₑ = 71.25x12/56.49 = 15.14 ksi
Since fₓ/Fₓ less than 0.15 use formula [H1-3, page 5-54 of Manual]
0.533/9.36 + 15.14/14.25 = 0.057 + 1.06 = 1.119

Member 7
Size W16x36, L = 4 ft, a = 10.6 in², Sz = 56.49 in³
L₂ = 7.37 ft (Clause F1-2 page 5-45 of Manual)
From observation load case 3 will govern, Fₓ = 24.06 kip (tension), Mₑ = 62.96 ft·kip
From Clause F1-1, the allowable compressive stress is
fₑ = 0.66 Fy = 23.76 ksi
Since section is in tension, per Clause F1-5 (page 5-45 of Manual)
fₑ = 0.60 · 36 = 21.60 Ksi
Choosing the larger of above 2 values, fₑ = 23.76 Ksi
fₑ = 24.06/10.6 = 2.2698
fₑ = 62.96 · 12/56.49 = 13.37
Since combined tension and bending, use equation H 2-1, page 5-55 of the AISC ASD 9th ed. specs.
2.2698/(0.6 · 36) + 13.37(23.76) = 0.105 + 0.5627 = 0.6677

Member 8
Size L4x4x1/4, L = 7.071 ft, a = 1.94 in²
From observation load case 1 will govern, Fₓ = 23.04 kip (Comp.)
Fa is computed as per page 5-310 of the AISC ASD 9th ed. specs.

\[ Q_s = 1.34 - 0.00447 \cdot (4/0.25) \cdot (36)^{1/2} = 0.9108 \]

\[ Q_a = 1.0 \]

\[ Q = Q_s \cdot Q_a = 0.9108 \]

\[ C_c = 92.0 \cdot \pi^2 \cdot E/(Q \cdot F_y)^{1/2} = [2.0 \cdot \pi^2 \cdot 29000/(0.9108 \cdot 36)]^{1/2} = 132.1241 \]

\[ K_I/r = 7.071 \cdot 12/0.795 = 106.73 < C_c. \]

Hence, \( f_a = 11.6027 \) ksi (computed per equation 4-1)

Actual compressive stress

\[ f_a = 23.04/1.94 = 11.876 \text{ ksi} \]

Therefore, Ratio =

\[ f_a/f_a = 11.876/11.602 = 1.024 \]

Member 9

Size L5x5x3/8, \( L = 5.657 \) ft, \( a = 3.61 \text{ in}^2 \)

From observation, load case 1 governs, \( F_x = 48.44 \) kip (Comp.)

Fa is computed as per page 5-310 of the AISC ASD 9th ed. specs.

\[ Q_s = 1.34 - 0.00447 \cdot (5/0.375) \cdot (36)^{1/2} = 0.9824 \]

\[ Q_a = 1.0 \]

\[ Q = Q_s \cdot Q_a = 0.9824 \]

\[ C_c = 92.0 \cdot \pi^2 \cdot E/(Q \cdot F_y)^{1/2} = [2.0 \cdot \pi^2 \cdot 29000/(0.9824 \cdot 36)]^{1/2} = 127.2238 \]

\[ (K_I/r)_{\text{min}} = 5.657 \cdot 12/0.99 = 68.57 < C_c. \]

Hence, \( f_a = 16.301 \) ksi (computed per equation 4-1)

Actual compressive stress

\[ f_a = 48.44/3.61 = 13.418 \text{ ksi} \]

Therefore Ratio =

\[ f_a/f_a = 13.418/16.301 = 0.823 \]

Comparison

<table>
<thead>
<tr>
<th>Governing stress ratio at member no.</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.158</td>
<td>1.154</td>
<td>negligible</td>
</tr>
<tr>
<td>2</td>
<td>0.914</td>
<td>0.913</td>
<td>negligible</td>
</tr>
<tr>
<td>3</td>
<td>1.117</td>
<td>1.121</td>
<td>negligible</td>
</tr>
<tr>
<td>4</td>
<td>0.936</td>
<td>0.939</td>
<td>negligible</td>
</tr>
<tr>
<td>5</td>
<td>0.579</td>
<td>0.580</td>
<td>negligible</td>
</tr>
</tbody>
</table>
### Governing stress ratio at member no.

<table>
<thead>
<tr>
<th>Member no.</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>1.119</td>
<td>1.107</td>
<td>1.1%</td>
</tr>
<tr>
<td>7</td>
<td>0.668</td>
<td>0.670</td>
<td>negligible</td>
</tr>
<tr>
<td>8</td>
<td>1.024</td>
<td>1.045</td>
<td>2.1%</td>
</tr>
<tr>
<td>9</td>
<td>0.823</td>
<td>0.815</td>
<td>1.0%</td>
</tr>
</tbody>
</table>

### STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\ASD 9th Edition\AISC ASD - 2D Frame validation.STD` is typically installed with the program.

**STAAD PLANE VERIFICATION PROBLEM NO 13**

**UNIT FEET KIP**

**JOIN COORDINATES**

1 0 0 0; 2 25 0 0; 3 0 10 0; 4 25 11 0; 5 0 15 0; 6 25 15 0; 7 5 15 0;
8 21 15 0;

**MEMBER INCIDENCES**

1 1 3; 2 3 5; 3 2 4; 4 4 6; 5 5 7; 6 7 8; 7 8 6; 8 3 7; 9 4 8;

**MEMBER PROPERTY AMERICAN**

1 2 TABLE ST W12X26
3 4 TABLE ST W14X43
5 TO 7 TABLE ST W16X36
8 TABLE ST L40404
9 TABLE ST L50506

**MEMBER TRUSS**

8 9

**DEFINE MATERIAL START**

**ISOTROPIC MATERIAL1**

E 4.176e+06
POISSON 0.3

**END DEFINE MATERIAL**

**CONSTANTS**

**MATERIAL MATERIAL1 ALL**

**SUPPORTS**

1 2 PINNED

**LOAD 1 DL + LL**

**MEMBER LOAD**

5 TO 7 UNI Y -2

**LOAD 2 WIND FROM LEFT**

**JOINT LOAD**

5 FX 15

**LOAD COMBINATION 3**

1 0.75 2 0.75
## Verification Examples

### V.09 Steel Design

```plaintext
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT MEMBER FORCES
PARAMETER 1
CODE AISC
TRACK 1 ALL
CHECK CODE ALL
FINISH

STAAD Output

---

STAAD.Pro CODE CHECKING - (AISC 9TH EDITION) v1.0

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

* 1 ST W12X26

FAIL AISC- H1-2 1.154 1
25.00 C 0.00 56.50 10.00

MEM= 1, UNIT KIP-INCH, L= 120.0 AX= 7.65 SZ= 33.4 SY= 5.3
KL/R-Y= 79.8 CB= 1.00 YLD= 36.00 ALLOWABLE STRESSES: FCZ= 20.21
FTZ= 21.60 FCY= 27.00 FTY= 27.00 FC= 15.38 FT= 21.60 FV= 14.40

---

2 ST W12X26

PASS AISC- H1-3 0.913 1
8.72 C 0.00 56.50 0.00

MEM= 2, UNIT KIP-INCH, L= 60.0 AX= 7.65 SZ= 33.4 SY= 5.3
KL/R-Y= 39.9 CB= 1.00 YLD= 36.00 ALLOWABLE STRESSES: FCZ= 23.76
FTZ= 23.76 FCY= 27.00 FTY= 27.00 FC= 19.20 FT= 21.60 FV= 14.40

---

* 3 ST W14X43

FAIL AISC- H1-3 1.121 3
25.50 C 0.00 -112.20 11.00

MEM= 3, UNIT KIP-INCH, L= 132.0 AX= 12.60 SZ= 62.5 SY= 11.3
KL/R-Y= 69.7 CB= 1.00 YLD= 36.00 ALLOWABLE STRESSES: FCZ= 21.60
FTZ= 21.60 FCY= 27.00 FTY= 27.00 FC= 16.46 FT= 21.60 FV= 14.40

---

4 ST W14X43

PASS AISC- H2-1 0.939 3
8.83 T 0.00 -112.20 0.00

MEM= 4, UNIT KIP-INCH, L= 48.0 AX= 12.60 SZ= 62.5 SY= 11.3
KL/R-Y= 25.3 CB= 1.00 YLD= 36.00 ALLOWABLE STRESSES: FCZ= 23.76
FTZ= 23.76 FCY= 27.00 FTY= 27.00 FC= 19.85 FT= 21.60 FV= 14.40

---

5 ST W16X36

PASS AISC- H1-3 0.580 3
14.02 C 0.00 -57.00 5.00

VERIFICATION PROBLEM NO 13 -- PAGE NO.

---

6

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)

MEMBER | TABLE | RESULT/ | CRITICAL COND/ | RATIO/ | LOADING/ |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
```

---

STAAD.Pro 4033 User Manual
V. Steel Design per AISC LRFD

V. AISC LRFD - Wide Flange Tension Capacity

To check the adequacy of a W8X24 American section to carry a tensile design load of 245 Kips.

Reference

Hand calculation using the following reference:

Problem
Assume the reduction in area due to bolt holes to be equivalent to a net section factor of 0.72.

\[ F_y = 50 \text{ ksi} \]
\[ L = 10 \text{ ft} \]

Hand Calculation
Net section = 0.72(7.08in.\(^2\)) = 5.11 in.\(^2\)

\[ \phi P_n = \frac{\phi t F_y A_g}{\phi t F_u A_e} = \frac{0.9(50\text{ksi})(7.08\text{in.}^2)}{0.75(65\text{ksi})(5.11\text{in.}^2)} = 318.6k \]
\[ \phi P_n = \frac{0.75(65\text{ksi})(5.11\text{in.}^2)}{0.75(65\text{ksi})(5.11\text{in.}^2)} = 248.5k \]

Comparison
The reported STAAD.Pro value is based on the term PNT in the STAAD.Pro output.

Table 569: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>248.5</td>
<td>248.51</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: The assumed NSF of 0.72 is comparable to the term Ae/Ag which evaluates to 5.11/7.08.
V. AISC LRFD - Angle Section Tension Capacity

To check the adequacy of a L4X4X1/2 American single angle to carry a tensile design load of 120 Kips.

Reference

Hand calculation using the following reference:


Problem

Assume the reduction in area due to bolt holes to be equivalent to a net section factor of 0.795.

\[ F_y = 36 \text{ ksi} \]

Hand Calculation

Net section \( = 0.795(3.75 \text{ in.}^2) = 2.98 \text{ in.}^2 \)

\[
\phi_P = \begin{cases} 
\phi_t F_y A_g = 0.9(36 \text{ksi})(3.75 \text{in.}^2) = 121.5k \\
\phi_t F_u A_e = 0.75(58 \text{ksi})(2.98 \text{in.}^2) = 129.6k 
\end{cases}
\]
Comparison

The reported STAAD.Pro value is based on the term PNT in the STAAD.Pro output.

Table 570: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>121.5</td>
<td>121.5</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: The assumed NSF of 0.795 is comparable to the term Ae/Ag which evaluates to 2.98/3.75.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Angle Section Tension Capacity.STD is typically installed with the program.

STAAD SPACE TENSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
* OBJECTIVE : CHECKING THE DESIGN AXIAL TENSILE STRENGTH OF A
* L4X4X1/2 AMERICAN SECTION. CAPACITY SHOULD BE 122 KIPS ACCORDING TO
* EXAMPLE 3-2 ON PAGE 3-7 OF 3RD EDITION LRFD CODE.
* UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST L40408
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 120
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FU 58 ALL
* NSF = 2.98/3.75 = 0.795
NSF 0.795 ALL
TRACK 1 ALL
V. AISC LRFD - Wide Flange Compression Capacity 1

To determine the capacity of W14X132 column in axial compression.

Reference

Hand calculation using the following reference:


Problem

The column is braced at its ends for both axes.

\[ F_y = 50 \text{ ksi} \]

\[ \text{Length} = 30 \text{ ft} \]

Hand Calculation

Net section = \( 0.795(3.75 \text{ in.}^2) = 2.98 \text{ in.}^2 \)

\[
\phi P_n = \left\{ \begin{array}{l} \phi_t F_y A_g = 0.9(36 \text{ksi})(3.75 \text{in.}^2) = 121.5 k \cr \phi_t F_u A_v = 0.75(58 \text{ksi})(2.98 \text{in.}^2) = 129.6 k \end{array} \right.
\]

Comparison

The reported STAAD.Pro value is based on the term PNT in the STAAD.Pro output.

Table 571: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>844</td>
<td>843.02</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Wide Flange Compression Capacity 1.STD is typically installed with the program.

STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION

*  EXAMPLE PROBLEM 4.1, CASE (A), PAGE 4-7, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 844 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -840
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 50 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ST</td>
<td>W14X132</td>
<td>PASS</td>
<td>LRFD-H1-1A-C</td>
<td>0.996</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>840.00 C</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION) v1.0
***************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST W14X132 (AISC SECTIONS) PASS LRFD-H1-1A-C 0.996 1
840.00 C 0.00 0.00 0.00
| DESIGN STRENGTHS FOR MEMBER 1 UNITS - KIP IN |
V. AISC LRFD - Wide Flange Compression Capacity 2

To determine the capacity of W14X132 column in axial compression. The column is braced at its ends for its major axis, and at ends and mid-height for the minor axis.

Reference

Hand calculation using the following reference:


Problem

\[ F_y = 50 \text{ ksi} \]

Length = 30 ft

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

Table 572: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>1,300</td>
<td>1,296.75</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Wide Flange Compression Capacity 2.STD is typically installed with the program.

STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.1, CASE (B), PAGE 4-7, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 1300 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
**V. AISC LRFD - Angle Section Compression Capacity**

To determine the design strength of an 8ft long single angle L4X3.5X5/16 in axial compression.

**Reference**

Hand calculation using the following reference:


**Problem**

The column is pinned at its ends and no intermediate bracing.

\[ F_y = 36 \text{ ksi} \]

Length = 8 ft

**Comparison**

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.
Table 573: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>28.7</td>
<td>28.66</td>
<td>Negligible</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Angle Section Compression Capacity.STD is typically installed with the program.

STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.3, PAGE 4-12, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 28.7 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 8 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST L40355
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -25
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 36 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION) v1.0
***************************************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
V. AISC LRFD - Tee Section Compression Capacity

To find a suitable American T-section with an axial compressive strength of 100 Kips.

Reference

Hand calculation using the following reference:


Problem

The member ends are pinned, and the member is braced at the ends only for both axes.

\[ F_y = 50 \text{ ksi} \]

\[ \text{Length} = 20 \text{ ft} \]

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

**Table 574: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>102</td>
<td>101.56</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Tee Section Compression Capacity.STD` is typically installed with the program.

**STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED**

**START JOB INFORMATION**

**ENGINEER DATE 23-Sep-18**

**END JOB INFORMATION**

* * EXAMPLE PROBLEM 4.4, PAGE 4-13, AISC LRFD 3RD ED. *
* * CAPACITY (PNC) SHOULD BE ABOUT 102 KIPS *

**UNIT FEET KIP**

**JOINT COORDINATES**

1 0 0 0; 2 0 20 0;

**MEMBER INCIDENCES**

1 1 2;
V. AISC LRFD - Rectangular HSS Compression Strength

To validate that an American ASTM A500 grade B HSS12X10X1/2 compression member has a design compressive strength of over 500 kips.

Reference

Hand calculation using the following reference:

_AISC Load Factor Resistance Design, 3rd Edition, Example 4.8, page 4-13 & Table 4-6, Page 4-50._

Problem

\[ F_y = 46 \text{ ksi} \]

\[ \text{Length} = 20 \text{ ft} \]
KY = KZ = 0.8

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

Table 575: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength in Axial Compression (kips)</td>
<td>634 (*)</td>
<td>634.26</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: *In the solved example, on page 4-14, the capacity is reported as ϕ×Pn = 580 kips. Looking at Table 4-6 on page 4-50, this happens to be the capacity for a 20 ft effective length, not a 16 ft effective length. As per the table, the capacity corresponding to a 16 ft effective length is 634 kips.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Rectangular HSS Compression Strength.STD is typically installed with the program.

STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
* * EXAMPLE PROBLEM 4.5, PAGE 4-13, AISC LRFD 3RD ED.
* * CAPACITY (PNC) SHOULD BE ABOUT 634 KIPS
* *
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST12X10X0.5
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -500
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
V. AISC LRFD - Double Angle Compression Capacity

To determine the design compressive strength of a long-leg-back-to-back double angle made of American L4X3.5X1/2 angles with 3/4 in. separation.

Reference

Hand calculation using the following reference:


Member is 8 ft long. Ends are pinned. Unbraced length should be assumed as the distance between ends.

Problem

\[ F_y = 36 \text{ ksi} \]

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

Table 576: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>101</td>
<td>101.22</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

STAAD Output
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Double Angle Compression Capacity.STD is typically installed with the program.

STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.7, PAGE 4-14, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 101 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 8 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TABLE LD L40355 SP 0.75
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JIONT LOAD
2 FY -100
PERFORM ANALYSIS
PARAMETER 1
CODE LRFD
FYLD 36 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION) v1.0
*******************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
     TABLE       FX     MY     MZ     LOCATION
=======================================================================
1  LD   L40355                   (AISC SECTIONS)
PASS     LRFD-H1-1A-C       0.988         1
100.00 C          0.00           0.00        0.00
+---------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP |
| PNC=  101.22  PNT=  145.80  MNZ=  81.97  MNY=  107.03  VN=  48.68 |
+---------------------------------------------------------------------+
To validate that an American Round HSS 10.000X0.5 compression member has a design compressive strength of about 364 kips.

Reference
Hand calculation using the following reference:
*AISC Load Factor Resistance Design*, 3rd Edition, Table 4-7, Page 4-68.

Problem
\[ F_y = 42 \text{ ksi} \]
\[ \text{Length} = 20 \text{ ft} \]

Comparison
The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

**Table 577: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design strength (kips)</td>
<td>364</td>
<td>364.22</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Round HSS Compression Capacity.STD` is typically installed with the program.
V. AISC LRFD - Wide Flange Flexural Strength 1

To determine the design flexural strength of an ASTM A992 W18X40 bent about its strong axis on a 35-ft span with a uniformly distributed load.

Reference

Hand calculation using the following reference:

*AISC Load Factor Resistance Design*, 3rd Edition, Example 5.1, Case (a), page 5-12.

Problem

The beam is braced such that $L_b = 2\ ft$. The deflection at mid-span due to a uniformly distributed service load of 1 kip/ft is also to be calculated.

$F_y = 50\ ksi$

$Length = 35\ ft$

Comparison

The reported STAAD.Pro value is based on the term $MNZ$ in the STAAD.Pro output.
### Table 578: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection (in)</td>
<td>1.90</td>
<td>1.8918</td>
<td>none</td>
</tr>
<tr>
<td>Design flexural strength</td>
<td>294</td>
<td>294</td>
<td>none</td>
</tr>
<tr>
<td>(ft-kips)</td>
<td></td>
<td>(3,528 in-kips)</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Wide Flange Flexural Strength 1.STD is typically installed with the program.

STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*  
*  EXAMPLE PROBLEM 5.1, CASE (A), PAGE 5-12, AISC LRFD 3RD ED.
  
  *  CAPACITY (MNZ) SHOULD BE ABOUT 294 KIP-FT
  
  *  
  UNIT FEET KIP
  JOINT COORDINATES
  1 0 0 0; 2 35 0 0;
  MEMBER INCIDENCES
  1 1 2;
  MEMBER PROPERTY AMERICAN
  1 TABLE ST W18X40
  UNIT FEET KIP
  DEFINE MATERIAL START
  ISOTROPIC MATERIAL1
  E 4.176e+06
  POISSON 0.3
  END DEFINE MATERIAL
  UNIT FEET KIP
  CONSTANTS
  MATERIAL MATERIAL1 ALL
  SUPPORTS
  1 PINNED
  2 FIXED BUT MZ
  LOAD 1
  MEMBER LOAD
  1 UNI GY -1
  LOAD COMBINATION 2
  1 1.8
  PERFORM ANALYSIS
  LOAD LIST 1
  SECTION 0.5 ALL
  PRINT SECTION DISPL
  LOAD LIST 2
  UNIT FEET KIP
  PARAMETER 1
  CODE LRFD
  MAIN 1 ALL
V. AISC LRFD - Wide Flange Flexural Strength 2

To determine the design flexural strength of an ASTM A992 W18X40 bent about its strong axis on a 35-ft span with a uniformly distributed load.

Reference
Hand calculation using the following reference:

* *AISC Load Factor Resistance Design*, 3rd Edition, Example 5.1, Case (b), page 5-12.

**Problem**

The beam is braced at the ends and at third points such that \( L_b = 11.7 \text{ ft} \).

\[
F_y = 50 \text{ ksi} \\
\text{Length} = 35 \text{ ft}
\]

**Comparison**

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

**Table 579: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design flexural strength</td>
<td>211</td>
<td>209.5</td>
<td>&lt;1%</td>
</tr>
<tr>
<td>(ft-kips)</td>
<td></td>
<td>(2,514 in-kips)</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** The STAAD.Pro result is based on a \( C_b \) of 1.0, while the theoretical solution is based on a \( C_b \) of 1.01

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Wide Flange Flexural Strength 2.STD is typically installed with the program.

STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED

START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*  
*  EXAMPLE PROBLEM 5.1, CASE (B), PAGE 5-12, AISC LRFD 3RD ED.
*  CAPACITY (MNZ) SHOULD BE ABOUT 211 KIP-FT
*  
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X40
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
**Verification Examples**

**V.09 Steel Design**

---

**STAAD Output**

<table>
<thead>
<tr>
<th>Y PROPERTIES</th>
<th>IN KIP</th>
<th>INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>MEMBER 1</td>
<td>AISC SECTIONS</td>
</tr>
<tr>
<td></td>
<td>ST W18X40</td>
<td></td>
</tr>
<tr>
<td></td>
<td>L1 L1 L1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>CAPACITIES</td>
<td>IN KIP</td>
</tr>
<tr>
<td></td>
<td>L1 L1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>IN KIP</td>
<td></td>
</tr>
<tr>
<td></td>
<td>INCH</td>
<td></td>
</tr>
<tr>
<td></td>
<td>L1 L1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>L1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>MNZ=0.2514E+4</td>
<td></td>
</tr>
<tr>
<td></td>
<td>MN=0.4283E+3</td>
<td></td>
</tr>
<tr>
<td></td>
<td>VN=0.1522E+3</td>
<td></td>
</tr>
<tr>
<td></td>
<td>VN=0.0000E+0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>RZ=0.7202E+1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>RY=0.1272E+1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PZ=0.7840E+2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PY=0.1000E+2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>AZ=0.4214E+1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FYLD=50.00</td>
<td></td>
</tr>
<tr>
<td></td>
<td>NSF=1.00</td>
<td></td>
</tr>
<tr>
<td></td>
<td>dff=0.00</td>
<td></td>
</tr>
<tr>
<td></td>
<td>ABSOLUTE MZ ENVELOPE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>WITH LOAD NO.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>max force/ moment summary (KIP- FEET)</td>
<td></td>
</tr>
</tbody>
</table>

**MAX FORCE/ MOMENT SUMMARY (KIP-FEET)**

<table>
<thead>
<tr>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>MOMENT-Y</th>
<th>MOMENT-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>VALUE</td>
<td>0.0</td>
<td>22.7</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOADING</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

---
V. AISC LRFD - Wide Flange Flexural Strength 3

To determine the design flexural strength of an ASTM A992 W18X40 bent about its strong axis on a 35-ft span with a uniformly distributed load.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 5.1, Case (c), page 5-12.

Problem

The beam is braced at the ends and so, $L_b = 35 \text{ ft}$.

$F_y = 50 \text{ ksi}$

Length = 35 ft

Comparison

The reported STAAD.Pro value is based on the term $MNZ$ in the STAAD.Pro output.

Table 580: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design flexural strength (ft·kips)</td>
<td>50.8</td>
<td>50.6</td>
<td>&lt;1%</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(607.2 in·kips)</td>
<td></td>
</tr>
</tbody>
</table>

Note: The $C_b$ of 1.14 calculated by STAAD.Pro precisely matches the value used in the reference hand calculation.
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X40
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -0.3
PERFORM ANALYSIS
PARAMETER 1
CODE LRFD
MAIN 1 ALL
CB 0 ALL
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>--- Y PROPERTIES</th>
<th>IN INCH UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER 1 AISC SECTIONS ST W18X40</td>
<td>AX=0.1180E+2</td>
</tr>
<tr>
<td>DESIGN CODE LRFD 2001</td>
<td>AZ=0.4214E+1</td>
</tr>
<tr>
<td>*</td>
<td>--Z</td>
</tr>
<tr>
<td></td>
<td>PZ=0.7840E+2</td>
</tr>
<tr>
<td></td>
<td>RY=0.1272E+1</td>
</tr>
<tr>
<td></td>
<td>RZ=0.7202E+1</td>
</tr>
<tr>
<td></td>
<td>45.9 (KIP-FEET)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>IN KIP INCH</th>
<th>L1 L1 L1</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>330.12</td>
<td>+ L1 L1</td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>58.32</td>
<td>+ L1 L1</td>
</tr>
<tr>
<td>UNL =</td>
<td>420.00</td>
<td>+ L1 L1</td>
</tr>
<tr>
<td>CB =</td>
<td>1.14</td>
<td>+ L1</td>
</tr>
<tr>
<td>PHIC =</td>
<td>0.85</td>
<td>+</td>
</tr>
<tr>
<td>PHIB =</td>
<td>0.90</td>
<td>+</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CAPACITIES</th>
<th>IN KIP INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>PNC=0.2310E+2</td>
<td></td>
</tr>
<tr>
<td>pnc=0.0000E+0</td>
<td></td>
</tr>
<tr>
<td>PNT=0.5310E+2</td>
<td></td>
</tr>
<tr>
<td>MNZ=0.6072E+3</td>
<td></td>
</tr>
<tr>
<td>mnz=0.5512E+3</td>
<td></td>
</tr>
</tbody>
</table>

Verification Examples
V.09 Steel Design

STAAD.Pro
4055 User Manual
V. AISC LRFD - Select Wide Flange 1

To select the optimum ASTM A992 W-shape with a design flexural strength of 500 kip-ft.

Reference


Problem

Beam is bent about its strong axis, and is continuously braced.

\[ F_y = 50 \text{ ksi} \]

\[ \text{Length} = 30 \text{ ft} \]

\[ w = 4.44 \text{ kip/ft} \text{ (equiv. of 500 ft·kip design moment)} \]

Comparison

*Table 581: Comparison of results*

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optimum section</td>
<td>W24x55</td>
<td>W24x55</td>
<td>none</td>
</tr>
<tr>
<td>Design strength, ( \phi_b M_z ) (ft·kips)</td>
<td>506</td>
<td>502.5 (*)</td>
<td>&lt;1%</td>
</tr>
</tbody>
</table>
* Calculated as MZ envelope value divided by the MZ ratio:

\[
499.95 \text{ ft-kips} / 0.995 = 502.5 \text{ ft-kips}
\]

**Note:** Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Select Wide Flange 1.STD is typically installed with the program.

STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
* * TABLE 5.2, PAGE 5-40, AISC LRFD 3RD ED.
* * OBJECTIVE : TO SELECT THE MOST OPTIMUM SECTION WITH A DESIGN FLEXURAL STRENGTH OF 500 KIP-FT.
* * FOR A FULLY BRACED BEAM, PER ABOVE REFERENCE, IT SHOULD BE A W24X55
* * 500 KIP-FT IS EQUAL TO AN APPLIED LOAD OF 4.444 KIP/FT ON A SIMPLY SUPPORTED BEAM
* * UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X26
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1 E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -4.444
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNT OF 1.0 INCH
UNT 1 ALL
MAIN 1 ALL
FYLD 50 ALL
CB 0 ALL
TRACK 2 ALL
### STAAD Output

**STAAD.PRO MEMBER SELECTION - (LRFD 3RD EDITION)  v1.0**

-------------------------------
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>DESIGN CODE</th>
<th>PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>LRFD 2001</td>
<td>INCH UNIT</td>
</tr>
<tr>
<td>ST W24X55</td>
<td></td>
<td>AX=0.1620E+2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>AZ=0.4720E+1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PY=0.1330E+2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PZ=0.1340E+3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>AX=0.1620E+2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>AZ=0.4720E+1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PY=0.1330E+2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PZ=0.1340E+3</td>
</tr>
</tbody>
</table>

**PARAMETER**

<table>
<thead>
<tr>
<th>IN KIP</th>
<th>INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>268.60</td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>39.44</td>
</tr>
<tr>
<td>UNL=</td>
<td>1.00</td>
</tr>
<tr>
<td>CB=</td>
<td>1.14</td>
</tr>
<tr>
<td>PHIC=</td>
<td>0.85</td>
</tr>
<tr>
<td>PHIB=</td>
<td>0.90</td>
</tr>
<tr>
<td>FYLD=</td>
<td>50.00</td>
</tr>
<tr>
<td>NSF=</td>
<td>0.00</td>
</tr>
<tr>
<td>DFF=</td>
<td>0.00</td>
</tr>
<tr>
<td>dff=</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**VALUE**

<table>
<thead>
<tr>
<th>VALUE</th>
<th>0.0</th>
<th>66.7</th>
<th>0.0</th>
<th>0.0</th>
<th>499.9</th>
</tr>
</thead>
</table>

**LOCATION**

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>15.0</th>
</tr>
</thead>
</table>

**MAX FORCE/ MOMENT SUMMARY**

<table>
<thead>
<tr>
<th>VALUE</th>
<th>0.0</th>
<th>66.7</th>
<th>0.0</th>
<th>0.0</th>
<th>499.9</th>
</tr>
</thead>
</table>

**LOADING**

<table>
<thead>
<tr>
<th>0.00</th>
<th>0.00</th>
<th>-499.95</th>
<th>15.00</th>
</tr>
</thead>
</table>

**RESULTS**

<table>
<thead>
<tr>
<th>RESULT/ CRITICAL</th>
<th>RATIO/ LOADING</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
</tr>
<tr>
<td>MZ</td>
<td>LOCATION</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PASS</th>
<th>LRFD-H1-1B-C</th>
<th>0.995</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>0.00</td>
<td>-499.95</td>
<td>15.00</td>
</tr>
</tbody>
</table>

**NOTE:**

- Verification Examples
- STAAD.Pro
- User Manual
V. AISC LRFD - Non Compact Wide Flange 1

To check the adequacy of a non-compact fully braced W shape to carry a uniformly distributed load of 5.5 kips/ft.

Reference


Problem

W14x90 shape. Beam is bent about its strong axis.

\[ F_y = 50 \text{ ksi} \]
\[ \text{Length} = 30 \text{ ft} \]

Hand Calculation

\[ M_N = 6,910 \text{ in} \cdot \text{kips} = wL^2/8 \]
\[ w = \frac{8M}{L^2} = \frac{8(6,910)}{(30 \cdot 12)^2} = 0.427 \text{kips/in} \]
\[ W = 0.427 \text{kips/in}(30 \text{ft})(12 \text{ in/ft}) = 153.6 \text{kips} \]
\[ \phi_b W_c = 153.6 \text{kips} (30 \text{ ft}) = 4,607 \text{ ft kips} \]

Comparison

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

Table 582: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum total Factored Load, W (kips)</td>
<td>154</td>
<td>153.6</td>
<td>none</td>
</tr>
<tr>
<td>( \phi_b W_c )</td>
<td>4,610</td>
<td>4,607</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Non Compact Wide Flange 1.STD is typically installed with the program.

STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
* TABLE 5-4, PAGE 5-65, AISC LRFD 3RD ED.
**OBJECTIVE:** TO CHECK THE ADEQUACY OF A NON-COMPACT W SHAPE.

* BEAM IS FULLY BRACED. ACCORDING TO ABOVE REFERENCE,
* CAPACITY OF A 30 FT LONG W14X90 IS 154/30=5.133 KIP/FT

UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -5.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNIT OF 1.0 INCH
UNT 1 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>AISC SECTIONS</th>
<th>ST W14X90</th>
<th>DESIGN CODE</th>
<th>LRFD 2001</th>
<th>&lt;-LENGTH (FT)= 30.00 ---&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
<td>LRFD 2001</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>CAPACITIES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>L1 L1 L1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Y</th>
<th>PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>IN INCH UNIT</td>
</tr>
<tr>
<td></td>
<td></td>
<td>--------------</td>
</tr>
</tbody>
</table>

* FULLY BRACED CONDITION IS ACHIEVED WITH A UNIT OF 1.0 INCH
UNT 1 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION) v1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>* -------------------------------</td>
</tr>
<tr>
<td>* ---------------------------------</td>
</tr>
<tr>
<td>* ---------------------------------</td>
</tr>
<tr>
<td>*</td>
</tr>
<tr>
<td>MEMBER 1 *</td>
</tr>
<tr>
<td>MEMBERS</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>CAPACITIES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>L1 L1 L1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Y</th>
<th>PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>IN INCH UNIT</td>
</tr>
<tr>
<td></td>
<td></td>
<td>--------------</td>
</tr>
</tbody>
</table>

* FULLY BRACED CONDITION IS ACHIEVED WITH A UNIT OF 1.0 INCH
UNT 1 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
V. AISC LRFD - Channel Shape Capacity

To check the adequacy of a fully braced C shape to carry a uniformly distributed load of 1.5 kips/ft.

Reference

*AISC Load Factor Resistance Design*, 3rd Edition, Table 5.9, page 5-120.

Problem

C12x25 shape. Beam is bent about its strong axis.

\[ F_y = 50 \text{ ksi} \]

Length = 20 ft

Hand Calculation

\[ MNZ = 952.6 \text{ in·kips} = \frac{wL^2}{8} \]
\[ w = \frac{8M}{l^2} = \frac{8(952.6)}{(20 \cdot 12)^2} = 0.132 \text{kips/in} \]

\[ W = 0.132 \text{kips/in}(20\text{ft})(12 \text{ in/ft}) = 31.8 \text{kips} \]

\[ \phi_b W_c = 31.8 \text{kips} (20 \text{ ft}) = 635.1 \text{ ft-kips} \]

Comparison

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

Table 583: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum total Factored Load, W (kips)</td>
<td>31.8</td>
<td>31.8</td>
<td>none</td>
</tr>
<tr>
<td>( \phi_b W_c )</td>
<td>635</td>
<td>635.1</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Channel Shape Capacity.STD is typically installed with the program.

STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
* TABLE 5-9, PAGE 5-120, AISC LRFD 3RD ED.
* OBJECTIVE: TO CHECK THE ADEQUACY OF A C SHAPE.
* BEAM IS FULLY BRACED. ACCORDING TO ABOVE REFERENCE,
* CAPACITY OF A 20 FT LONG C12X25 IS 31.8/20=1.59 KIP/FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST C12X25
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
**Verification Examples**

**V.09 Steel Design**

```plaintext
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -1.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNT OF 1.0 INCH
UNT 1 ALL
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

---

**STAAD Output**

```

```

```

```

```
V. AISC LRFD - MC Shape Capacity

To check the adequacy of a fully braced MC shape to carry a uniformly distributed load of 1.34 kips/ft.

Reference

Problem
MC10x41.1 shape. Beam is bent about its strong axis.

\[
F_y = 50 \text{ ksi} \\
\text{Length} = 25 \text{ ft}
\]

Hand Calculation

\[
MNZ = 1,273 \text{ in} \cdot \text{kips} = \frac{wl^2}{8}
\]

\[
w = \frac{8M}{l^2} = \frac{8(1,273)}{(25 \cdot 12)^2} = 0.113 \text{kips/in}
\]

\[
W = 0.113 \text{kips/in}(25 \text{in})(12 \text{ in/ft}) = 33.9 \text{kips}
\]

\[
\phi_b W_c = 33.9 \text{kips}(25 \text{ft}) = 849 \text{ ft kips}
\]

Comparison

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

Table 584: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum total Factored Load, W (kips)</td>
<td>34</td>
<td>33.9</td>
<td>none</td>
</tr>
<tr>
<td>(\phi_b W_c)</td>
<td>849</td>
<td>849</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - MC Shape Capacity.STD is typically installed with the program.

STAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
* TABLE 5-10, PAGE 5-124, AISC LRFD 3RD ED.
* OBJECTIVE : TO CHECK THE ADEQUACY OF AN MC SHAPE.
* BEAM IS FULLY BRACED. ACCORDING TO ABOVE REFERENCE,
* CAPACITY OF A 25 FT LONG MC10X41 IS APPROX. 34/25=1.36 KIP/FT
* 
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST MC10X41
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -1.34
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNT OF 1.0 INCH
UNT 1 ALL
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>Y</th>
<th>PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>===</td>
<td>----------</td>
</tr>
</tbody>
</table>
V. AISC LRFD - Non Compact Wide Flange 2

To determine the design bending strength of a non-compact American wide flange section bent about its strong axis. The deflection at mid-span due to 2 concentrated loads acting at the third points along the beam is also to be calculated.

Reference

Problem

W21x48 shape

\[ F_y = 50 \text{ ksi} \]

Length = 40 ft

\[ L_b = 5 \text{ ft} \]

Comparison

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

**Table 585: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection (in)</td>
<td>1.41</td>
<td>1.41</td>
<td>none</td>
</tr>
<tr>
<td>Design bending strength, ( \phi_b M_n )</td>
<td>398</td>
<td>398.4</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(4,780.24 in·kips)</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Non Compact Wide Flange 2.STD is typically installed with the program.

**STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED**

**START JOB INFORMATION**

**ENGINEER DATE 23-Sep-18**

**END JOB INFORMATION**

* * EXAMPLE PROBLEM 5.2, CASE (A), PAGE 5-16, AISC LRFD 3RD ED.
* * CAPACITY (MNZ) SHOULD BE ABOUT 398 KIP-FT

* * UNIT FEET KIP

**JOINT COORDINATES**

1 0 0 0; 2 40 0 0;

* *

**MEMBER INCIDENCES**

1 1 2;

* *

**MEMBER PROPERTY AMERICAN**

1 TABLE ST W21X48

* *

**DEFINE MATERIAL START**

**ISOTROPIC MATERIAL1**

E 4.176e+06

POISSON 0.3

**END DEFINE MATERIAL**

**CONSTANTS**

**MATERIAL MATERIAL1 ALL**

* *
SUPPORTS
1 PINNED
2 FIXED BUT MZ
*
LOAD 1
MEMBER LOAD
1 CON GY -10 13.333
1 CON GY -10 26.667
*
PERFORM ANALYSIS
*
SECTION 0.5 ALL
PRINT SECTION DISPL
*
PARAMETER 1
CODE LRFD
MAIN 1 ALL
FYLD 7200 ALL
UNT 5 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH

STAAD Output

MEMBER SECTION DISPLACEMENTS
---------------------------------
UNITS ARE - INCH
MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
1     1       0.0000    0.0000    0.0000    0.0000   -0.3644    0.0000
0.0000   -0.7057    0.0000    0.0000   -1.0010    0.0000
0.0000   -1.2273    0.0000    0.0000   -1.3654    0.0000
0.0000   -1.4114    0.0000    0.0000   -1.3654    0.0000
0.0000   -1.2273    0.0000    0.0000   -1.0010    0.0000
0.0000   -0.7057    0.0000    0.0000   -0.3644    0.0000
0.0000    0.0000    0.0000
MAX LOCAL  DISP =    1.41141   AT     240.00  LOAD    1   L/DISP=    340
************ END OF SECT DISPL RESULTS ************
V. AISC LRFD - Non Compact Wide Flange 3

To determine the design bending strength of a non-compact American wide flange section bent about its strong axis.

Reference


Problem

W21x48 shape

\[ F_y = 50 \text{ ksi} \]

Length = 40 ft

\[ L_b = 13.3 \text{ ft} \]

Comparison

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

Table 586: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design bending strength, ( \phi_b M_n )</td>
<td>306</td>
<td>304.9</td>
<td>&lt;1%</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(3,658.52 in-kips)</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Non Compact Wide Flange 3.STD is typically installed with the program.

STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*EXAMPLE PROBLEM 5.2, CASE (B), PAGE 5-16, AISC LRFD 3RD ED.
*CAPACITY (MNZ) SHOULD BE ABOUT 306 KIP-FT
*UNIT FEET KIP
JOIN COORDINATES
1 0 0 0; 2 40 0 0;
*MEMBER INCIDENCES
1 1 2;  
V. AISC LRFD - Wide Flange Compression Capacity 4

To determine the design bending strength of a non-compact American wide flange section bent about its strong axis.

Reference


Problem
W21x48 shape

\[ F_y = 50 \text{ ksi} \]

Length = 40 ft

\[ L_b = 40 \text{ ft (braced at end points only)} \]

Comparison

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

**Table 587: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design bending strength, ( \phi_b M_n )</td>
<td>70.2</td>
<td>69.53</td>
<td>&lt;1%</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(834.4 in-kips)</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** The \( C_b \) of 1.14 calculated by STAAD.Pro precisely matches the value used in the reference hand calculation.

**STAAD Input**

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Wide Flange Compression Capacity 4.STD` is typically installed with the program.

**START JOB INFORMATION**

**ENGINEER DATE 23-Sep-18**

**END JOB INFORMATION**

* EXAMPLE PROBLEM 5.2, CASE (C), PAGE 5-16, AISC LRFD 3RD ED.
* CAPACITY (MNZ) SHOULD BE ABOUT 70.2 KIP-FT

**UNIT FEET KIP**

**JOINT COORDINATES**

1 0 0 0; 2 40 0 0;

* **MEMBER INCIDENCES**

1 1 2;

* **MEMBER PROPERTY AMERICAN**

1 TABLE ST W21X48

* **DEFINE MATERIAL START**

ISOTROPIC MATERIAL1

E 4.176e+06

POISSON 0.3

**END DEFINE MATERIAL**

**CONSTANTS**

**MATERIAL MATERIAL1 ALL**

* **SUPPORTS**

1 PINNED

2 FIXED BUT MZ

*
### Parameter 1

- **Code**: LRFD
- **Main**: ALL
- **FYLD**: 7200 ALL
- **CB**: 0 ALL
- **Track**: 2 ALL
- **Check Code**: ALL
- **Finish**

**STAAD Output**

#### STAAD.PRO Code Checking - (LRFD 3rd Edition) v1.0

<table>
<thead>
<tr>
<th>MEMBER 1</th>
<th>AISC Sections</th>
<th>ST W21X48</th>
<th>ST AISC W21X48</th>
<th>ST W21X48</th>
</tr>
</thead>
<tbody>
<tr>
<td>DESIGN CODE</td>
<td>LRFD 2001</td>
<td>ST AISC W21X48</td>
<td>ST AISC W21X48</td>
<td>ST W21X48</td>
</tr>
<tr>
<td>LENGTH (FT)</td>
<td>40.00</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CAPACITIES</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>IN KIP</td>
<td>IN INCH</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PNC</td>
<td>0.3584E+2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MNZ</td>
<td>0.8344E+3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MNZ</td>
<td>0.1600E+4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>VN</td>
<td>0.1947E+3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>VN</td>
<td>0.1000E+2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MAX FORCE/ MOMENT SUMMARY (KIP-FEET)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>AXIAL</td>
<td>SHEAR-Y</td>
<td>SHEAR-Z</td>
<td>MOMENT-Y</td>
<td>MOMENT-Z</td>
</tr>
<tr>
<td>VALUE</td>
<td>0.0</td>
<td>10.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOADING</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
V. AISC LRFD - Select Wide Flange 2

To find the optimum W shape with a flexural design strength of 150 ft-kips. Beam is braced at ends and at mid-span point.

Reference


Problem

The beam should also satisfy a deflection limit of 1.0 inches under service load.

\[ F_y = 50 \text{ ksi} \]
\[ \text{Length} = 20 \text{ ft} \]
\[ L_b = 10 \text{ ft} \]

Service load of 2.0 kip/ft. Deflection limit under service load = 1.0 in.

Comparison

The reported STAAD.Pro value is based on the term MNZ in the STAAD.Pro output.

Table 588: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optimum Section</td>
<td>W16x31</td>
<td>W16x31</td>
<td>none</td>
</tr>
<tr>
<td>Design bending strength, ( \phi_b M_n )</td>
<td>150</td>
<td>150.17</td>
<td>negligible</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(1,802 in-kips)</td>
<td></td>
</tr>
<tr>
<td>Maximum permissible deflection (in)</td>
<td>1.0</td>
<td>0.658</td>
<td>N/A (*)</td>
</tr>
</tbody>
</table>

Note: (*) The STAAD.Pro reported maximum deflection is less than 1.0 in; therefore this section satisfies the requirement. However, this value is not to be directly compared with the limit itself.
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Select Wide Flange 2.STD is typically installed with the program.

STAAD SPACE BENDING PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
* EXAMPLE 5.4, PAGE 5-21, AISC LRFD 3RD ED.
* OBJECTIVE: TO FIND THE OPTIMUM W SHAPE WITH A FLEXURAL
* DESIGN STRENGTH OF 150 KIP-FT.
*
* ACCORDING TO ABOVE REFERENCE, THE SECTION SHOULD BE W16X31
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X26
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -2
LOAD COMBINATION 2
1 1.5
PERFORM ANALYSIS
LOAD LIST 2
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
UNT 120 ALL
FYLD 50 ALL
TRACK 1 ALL
SELECT ALL
LOAD LIST ALL
PERFORM ANALYSIS
LOAD LIST 1
SECTION 0.5 ALL
PRINT SECTION DISPL
LOAD LIST 2
CHECK CODE ALL
FINISH
V. AISC LRFD - Load Capacity of 3 Wide Flange Beams

To find the design flexural strength capacity of 3 beams of different spans using the data in Table 5-4 of AISC LRFD 3rd edition.

Reference


Problem

W21x62 section.

\[ F_y = 50 \text{ ksi} \]

40 ft, 30 ft, and 8 ft spans are investigated. All beams are fully braced.

Comparison

The reported STAAD.Pro value is based on the ratio of \( MNZ \) in the STAAD output.

Table 589: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Utilization ratio, 30 ft span</td>
<td>1.0</td>
<td>1.0</td>
<td>none</td>
</tr>
<tr>
<td>Utilization ratio, 20 ft span</td>
<td>1.0</td>
<td>1.0</td>
<td>none</td>
</tr>
<tr>
<td>Utilization ratio, 8 ft span</td>
<td>1.0</td>
<td>1.001</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Load Capacity of 3 Wide Flange Beams.STD is typically installed with the program.
MEMBER PROPERTY AMERICAN
1 TO 3 TABLE ST W21X62
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 3 5 PINNED
2 4 6 FIXED BUT MZ
* 
LOAD 1
MEMBER LOAD
* ON MEMBER 1, W=108/40=2.7
1 UNI GY -2.7
* ON MEMBER 2, W=144/30
2 UNI GY -4.8
* ON MEMBER 3, W=454/8
3 UNI GY -56.75
* 
PERFORM ANALYSIS
PARAMETER 1
CODE LRFD
MAIN 1 ALL
FYLD 7200 ALL
* FULLY BRACED CONDITION CAN BE ACHIEVED BY SETTING UNT TO 1.0 INCH
UNT 0.1 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION) v1.0
*****************************************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
*    1  ST    W21X62       (AISC SECTIONS)
         FAIL    LRFD-H1-1B-C   1.000   1
         0.00 C  0.00        -540.00     20.00
+---------------------------------------------------------------------+
   | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN |
   | PNC=  53.25  PNT=  823.50  MNZ= 6480.00  MNY=  942.05  VN=  226.80 |
+---------------------------------------------------------------------+
*    2  ST    W21X62       (AISC SECTIONS)
         PASS    LRFD-H1-1B-C   1.000   1
         0.00 C  0.00        -540.00     15.00
+---------------------------------------------------------------------+
   | DESIGN STRENGTHS FOR MEMBER     2       UNITS - KIP IN |
   | PNC=  94.66  PNT=  823.50  MNZ= 6480.00  MNY=  942.05  VN=  226.80 |
+---------------------------------------------------------------------+
*    3  ST    W21X62       (AISC SECTIONS)
         FAIL    SHEAR-Y    1.001   1
V. AISC LRFD - Tension and Strong Axis Bending

To check the adequacy of an ASTM A992 W10X22 subject to axial tension and flexure about the strong axis.

Reference


Problem

W10x22 section.

\[ F_y = 50 \text{ ksi} \]
\[ F_u = 65 \text{ ksi} \]
\[ P_u = 55 \text{ kips} \]
\[ M_{uy} = 0, M_{uz} = 55 \text{ ft·kips} \]
\[ L_b = 4 \text{ ft} \]

Comparison

The reported STAAD.Pro value is based on the terms PNT and MNZ in the STAAD.Pro output.

Table 590: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design tensile strength, ( \phi_t P_n ) (kips)</td>
<td>292.0</td>
<td>292</td>
<td>none</td>
</tr>
<tr>
<td>Design bending strength, ( \phi_b M_{ny} ) (ft·kips)</td>
<td>97.5</td>
<td>97.5</td>
<td>none</td>
</tr>
<tr>
<td>Utilization ratio</td>
<td>0.98</td>
<td>0.98</td>
<td>none</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Note: Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Tension and Strong Axis Bending.STD is typically installed with the program.

STAAD SPACE COMBINED AXIAL TENSION + FLEXURE PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
EXAMPLE PROBLEM 6.1, CASE (B), PAGE 6-5, AISC LRFD 3RD ED.
INTERACTION RATIO SHOULD BE ABOUT 0.98

UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W10X22
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 140
2 MZ 55
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 50 ALL
FU 65 ALL
UNT 48 ALL
UNB 48 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>AISC SECTIONS</th>
<th>ST W10X22</th>
<th>Ax=0.6490E+1</th>
</tr>
</thead>
<tbody>
<tr>
<td>DESIGN CODE</td>
<td>LRFD 2001</td>
<td>-Z AV=0.2448E+1</td>
<td></td>
</tr>
<tr>
<td>*</td>
<td>LENGTH (FT)= 10.00 ---&gt;</td>
<td>AV=0.6100E+1</td>
<td></td>
</tr>
<tr>
<td>PARAMETER</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>IN KIP INCH</td>
<td></td>
<td>L1 L1 L1</td>
<td></td>
</tr>
</tbody>
</table>

Verification Examples
V.09 Steel Design
STAAD.Pro
4078
User Manual
V. AISC LRFD - Compression and Biaxial Bending

To check the adequacy of an ASTM A992 W14X176 beam subject to axial compression and biaxial bending.

Reference


Problem

W14x176 section.

\[ F_y = 50 \text{ ksi} \]
\[ P_u = 1,400 \text{ kips} \]
\[ M_{uy} = 70 \text{ ft·kips}, M_{uz} = 200 \text{ ft·kips} \]
\[ L_b = 14 \text{ ft} \]

Reverse curvature bending with equal end moments for both axis.

Hand Calculations
According to page 16.1 - 97 of the LRFD 3rd ed. code,

\[ M_p = F_y \cdot Z \leq 1.5F_y \cdot S \]

For a W14x176,

\[ 0.9 (1.5F_y \cdot S) = 0.9 [1.5 (50)(107)] = 7,222 \text{ kip-in} = 601.8 \text{ kip-ft}, \]

which is lower than \((0.9 F_y \cdot Z) = 611\). Hence, STAAD.Pro uses this limit state.

**Comparison**

The reported STAAD.Pro value is based on the terms PNC, MNY, and MNZ in the STAAD.Pro output.

**Table 591: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design compressive strength, (\phi_p \cdot P_n) (kips)</td>
<td>1,940</td>
<td>1,938</td>
<td>negligible</td>
</tr>
<tr>
<td>Design bending strength, (\phi_b M_{ny}) (ft-kips)</td>
<td>611</td>
<td>600.5</td>
<td>1.7%</td>
</tr>
<tr>
<td>Design bending strength, (\phi_b M_{nz}) (ft-kips)</td>
<td>1,200</td>
<td>1,200</td>
<td>none</td>
</tr>
<tr>
<td>Utilization ratio</td>
<td>0.973</td>
<td>0.974</td>
<td>negligible</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Compression and Biaxial Bending.STD is typically installed with the program.

STAAD SPACE AXIAL COMPRESSION + BIAXIAL BENDING PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*  EXAMPLE PROBLEM 6.2, PAGE 6-6, AISC LRFD 3RD ED.
*  INTERACTION RATIO SHOULD BE ABOUT 0.973
*  UNIT FEET KIP
JOIN COORDINATES
1 0 0 0; 2 14 0 0;
*MEMBER INCIDENCES
1 1 2;
*MEMBER PROPERTY AMERICAN
1 TABLE ST W14X176
*DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
*
SUPPORTS
1 FIXED BUT FX MX MY MZ
2 FIXED BUT MY MZ
*
LOAD 1
JOINT LOAD
1 FX 1400
*
LOAD 2
JOINT LOAD
1 MZ 200
2 MZ 200
*
LOAD 3
JOINT LOAD
1 MY 70
2 MY 70
*
LOAD COMBINATION 4
1 1.0 2 1.0 3 1.0
*
PERFORM ANALYSIS
LOAD LIST 4
PRINT MEMBER FORCES
*
PARAMETER 1
CODE LRFD
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
*
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>Member 1</th>
<th>AISC Sections</th>
<th>ST</th>
<th>W14X176</th>
<th>Design Code</th>
<th>LRFD 2001</th>
<th>Length (FT)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>AISC Sections</td>
<td>ST</td>
<td>W14X176</td>
<td></td>
<td>14.00</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>L4</th>
<th>CAPACITIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>IN KIP INCH</td>
<td>L4</td>
<td>IN KIP INCH</td>
</tr>
</tbody>
</table>
**V. AISC LRFD - Select Compression and Biaxial Bending**

To select the lightest ASTM A992 W14 section capable of carrying an axial compression plus biaxial bending.

**Reference**

**Problem**

\[ F_y = 50 \text{ ksi} \]

\[ P_u = 400 \text{ kips} \]

\[ M_{uy} = 80 \text{ ft-kips}, \ M_{uz} = 250 \text{ ft-kips} \]

\[ L_b = 14 \text{ ft} \]

Reverse curvature bending with equal end moments for both axis.

**Comparison**
Table 592: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selection section</td>
<td>W14x99</td>
<td>W14x99</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\LRFD 3rd Edition\AISC LRFD - Select Compression and Biaxial Bending.STD is typically installed with the program.

STAAD SPACE AXIAL COMPRESSION + BIAXIAL BENDING PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 6.4, PAGE 6-9, AISC LRFD 3RD ED.
*  THE LIGHTEST SUITABLE SECTION SHOULD BE A W14X99
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 14 0 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
*
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
*
SUPPORTS
1 FIXED
*
LOAD 1
JOINT LOAD
2 FX -400
2 MZ 250
2 MY 80
*
PERFORM ANALYSIS
*
PARAMETER 1
CODE LRFD
FYLD 7200 ALL
TRACK 2 ALL
SELECT ALL
**Verification Examples**

*FINISH*

---

STAAD Output

**AXIAL COMPRESSION + BIAXIAL BENDING PER AISC LRFD 3RD ED -- PAGE NO. 3**

---

STAAD.PRO MEMBER SELECTION - (LRFD 3RD EDITION) v1.0

---

**------------------------------**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>AISC SECTIONS</th>
<th>DESIGN CODE</th>
<th>LRFD 2001</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ST W14X99</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**-------------------------------**

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>VALUE</th>
<th>LOCATION</th>
<th>LOADING</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y=</td>
<td>45.20</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>KL/R-Z=</td>
<td>27.20</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>UNL=</td>
<td>168.00</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>CB=</td>
<td>1.00</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>PHIC=</td>
<td>0.85</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>PHIB=</td>
<td>0.90</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>FYLD=</td>
<td>50.00</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>NSF=</td>
<td>1.00</td>
<td>0.0</td>
<td>1</td>
</tr>
<tr>
<td>DFF=</td>
<td>0.00</td>
<td>0.0</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**-------------------------------**

<table>
<thead>
<tr>
<th>VALUE</th>
<th>400.0</th>
<th>0.0</th>
<th>0.0</th>
<th>80.0</th>
<th>250.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOADING</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**-------------------------------**

**MAX FORCE/ MOMENT SUMMARY**

**-------------------------------**

<table>
<thead>
<tr>
<th>VALUE</th>
<th>400.0</th>
<th>0.0</th>
<th>0.0</th>
<th>80.0</th>
<th>250.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>LOADING</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**-------------------------------**

<table>
<thead>
<tr>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
</tr>
<tr>
<td>PASS</td>
</tr>
<tr>
<td>400.00 C</td>
</tr>
</tbody>
</table>
V.AISC N690

V.AISC N690 1994

V. AISC N690 1994 Angle
Verify the allowable stress and critical ratio of the brace members in a frame using the AISC N690 1994 code.

Details

The horizontal load is 7.5 kips and the vertical load is 10 kips.

The forces due to loading in member 6 is 2.861 kips (tension) and in member 7 is 7.328 kips (compression).

Members 6 and 7 are "truss" members (axial-only).

Both member 6 and 7 are L 80x60x10, austenitic stainless steel ($F_y = 36$ ksi, $F_u = 58$ ksi).

Assume a net section factor of 0.9 and an effective length factor of 0.85.

Validation

\[ L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in} \]

Slenderness ratio (same for both members) using the minimum value of \( r \):

For member 6, \( kL/r_2 = 0.85(93.726)/1.29 = 61.76 \)

For member 7, \( kL/r_2 = 61.76 < 300 \)

For member 7, \( kL/r_2 = 61.76 < 200 \)
For members in compression (member 7), the equivalent slenderness ratio per Eqn. 4-4 of AISC ASD p5-311 must also be determined:

\[(KL/r)_{\text{equiv}} = \pi \times \sqrt{(E/F_e)}\]

where \(F_e = \) the elastic buckling strength for flexural torsional buckling taken to be 60.348 ksi. This is defined as the lowest root of the Eqn in AISC ASD (C4-2) page 5-317.

Thus, \((KL/r)_{\text{equiv}} = \pi \times \sqrt{(29,000/60.348)} = 68.86\)

**Axial Tension: Member 6**

Actual tensile stress (gross) = \(P/A = 2.862 / 8.41 = 0.340\) ksi

Allowable tensile stress (Tension capacity) = \(0.6 \times F_y = 0.6 \times 36 = 21.6\) ksi

Critical ratio = \(0.340/21.6 = 0.016\)

**Axial Compression: Member 7**

Actual compressive stress = \(P/A = 7.328 / 8.41 = 0.871\) ksi

As this member is made from austenitic stainless steel the allowable compression stress should be determined according to clause Q.1.5.9. As \(KL/r < 120\), then the value of allowable axial stress, \(F_a\), is determined from Eqn. Q1.5-11:

\[F_a = \frac{F_y}{2.15} \cdot \left(\frac{F_y}{2.15} \cdot \frac{6}{120}\right) \times KL \bigg| r = 16.74 \cdot \left(\frac{16.71 \cdot 6}{120}\right) 68.86 = 10.57\) ksi

Check cross-section meets requirements of Q1.9, which sets a limit for any unstiffened element clause Q1.9.1.2 of \(75/\sqrt{(F_y)} = 12.67\).

- Short leg = \(B/t = 6/0.625 = 9.5 < 12.67\), OK
- Long leg = \(D/t = 8/0.625 = 12.8 > 12.67\), leg is slender

Since \(75/\sqrt{(F_y)} = 12.67 < D/t = 12.8 < 155/\sqrt{(F_y)} = 25.83\), \(Q_s\) is determined by Eqn. QC2-1:

\[Q_s = 1.340 \times 0.00447 \times (b/t) \times \sqrt{(F_y)} = 0.9967\]

Allowable compressive stress, \(F_a = 0.9967 \times 10.57 = 10.54\) ksi

Critical ratio = \(0.871 / 10.54 = 0.083\)

Results

**Table 593: Verification comparison for AISC_N690_1994_Angle**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member 6 (tension)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (ksi)</td>
<td>340</td>
<td>340</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
## Parameter Table

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Member 7 (compression)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Critical slenderness ratio</td>
<td>68.86</td>
<td>68.87</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (ksi)</td>
<td>0.871</td>
<td>0.87</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable compressive stress (ksi)</td>
<td>10.54</td>
<td>10.55</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.083</td>
<td>0.083</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

## STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1994\AISC N690 1994 Angle.STD` is typically installed with the program.

### START JOB INFORMATION
- ENGINEER DATE 26-Aug-18

### JOINT COORDINATES
- 1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;

### MEMBER INCIDENCES
- 1 1 4;
- 2 4 6;
- 3 6 5;
- 4 5 3;
- 5 6 2;
- 6 1 6;
- 7 6 3;

### DEFINE MATERIAL ISOTROPIC STEEL
- E 2.9e+07
- POISSON 0.3
- DENSITY 0.283
- ALPHA 1.2e-05
- DAMP 0.03

### MEMBER PROPERTY AMERICAN
- 1 TO 5 TABLE ST W8X48
- 6 7 TABLE ST L806010

### MATERIALS
- STEEL ALL

### SUPPORTS
- 1 TO 3 FIXED

### MEMBER TRUSS
- 6 7

### PRINT MEMBER PROPERTIES all

### LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
************************************************************
PARAMETER 1
CODE AISC N690 1994
* Material strength
FU 58000 MEMB 6 7
FYLD 36000 MEMB 6 7
* Net area reduced by 10%
NSF 0.9 MEMB 6 7
* Austenitic stainless steel
STYPE 1 MEMB 6 7
* Effective length factors 0.85 in both axes
LX 1 MEMB 6 7
KY 0.85 MEMB 6 7
KZ 0.85 MEMB 6 7
*
TRACK 2 ALL
CHECK CODE MEMB 6 7
FINISH

STAAD Output

|--------------------------------------------------------------------------|
|                                                    Y
| PROPERTIES   |*************                                       |        IN INCH
| UNIT |            *  |=============================|   ==| |==
| 8.41 | |MEMBER 6 *  |  AISC SECTIONS              |     | |  --Z  AY =
| 3.33 | |            *  | ST L806010                 |     | |       AZ =
| 2.50 | |DESIGN CODE * |                             |     | |       AX =
| 11.99 | |ANSI N690-94* ===============================   ==| |==
| 5.16 | |            *                                                SZ =
| 2.81 | |<---LENGTH (FT) =  7.81 --->|               RY =
| 1.29 | |                      ************
| | |     0.0 (KIP- FEET) |
| | |
### Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(KL/R-Y)</td>
<td>28.34</td>
</tr>
<tr>
<td>(KL/R-Z)</td>
<td>61.76</td>
</tr>
<tr>
<td>(UNL)</td>
<td>93.72</td>
</tr>
<tr>
<td>(CMY)</td>
<td>0.00</td>
</tr>
<tr>
<td>(CMZ)</td>
<td>0.00</td>
</tr>
<tr>
<td>(FYLD)</td>
<td>36.00</td>
</tr>
<tr>
<td>(NSF)</td>
<td>0.90</td>
</tr>
<tr>
<td>(DFF)</td>
<td>0.00</td>
</tr>
<tr>
<td>((KL/R)_{\text{max}})</td>
<td>68.87</td>
</tr>
</tbody>
</table>

### Stresses

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(FA)</td>
<td>21.60</td>
</tr>
<tr>
<td>(fa)</td>
<td>0.34</td>
</tr>
<tr>
<td>(FCZ)</td>
<td>0.00</td>
</tr>
<tr>
<td>(FTZ)</td>
<td>0.00</td>
</tr>
<tr>
<td>(FCY)</td>
<td>0.00</td>
</tr>
<tr>
<td>(FTY)</td>
<td>0.00</td>
</tr>
<tr>
<td>(fbz)</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Force/Moment Summary (KIP·FEET)

<table>
<thead>
<tr>
<th>Location</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Moment-Y</th>
<th>Moment-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>-2.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### Design Summary (KIP·FEET)

* Verification Examples
  * V.09 Steel Design
    * STAAD.Pro 4089 User Manual
Verification Examples

V.09 Steel Design

* | RESULT/    CRITICAL COND/    RATIO/ |
LOADING/ *| FX         MY         MZ |
LOCATION |
=======================================================================
| PASS        ANSI Q1.6-2 0.016 |
1 | 2.86 T 0.00 0.00 |
0.00 |
**************************************************************************

STAAD SPACE -- PAGE NO. 5

ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)

STAAD SPACE -- PAGE NO. 6

STAAD.PRO CODE CHECKING - ( ANSI N690-1994) v1.0
**************************************************************************

| PROPERTIES
| **********
| UNIT |
| **==|==
| "=" |
| **==|==
| "=" |
| **==|==
| **==|==
| AX = |
| ST L806010 |
| --Z AY = |
| AZ = |
| == SY = |
| SZ = |

STAAD.Pro 4090 User Manual
## Verification Examples

V.09 Steel Design

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>L0 IN KIP</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Y</td>
<td>28.34</td>
</tr>
<tr>
<td>KL/R-Z</td>
<td>61.76</td>
</tr>
<tr>
<td>UNL</td>
<td>93.72</td>
</tr>
<tr>
<td>CB</td>
<td>0.00</td>
</tr>
<tr>
<td>CMY</td>
<td>0.00</td>
</tr>
<tr>
<td>CMZ</td>
<td>0.00</td>
</tr>
<tr>
<td>FYLD</td>
<td>36.00</td>
</tr>
<tr>
<td>NSF</td>
<td>0.90</td>
</tr>
<tr>
<td>DFF</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Absolute MZ Envelope**

| (KL/R)max   | 68.87     |

### MAX FORCE/ MOMENT SUMMARY (KIP- FEET)

| VALUE   | 7.3  |
| LOCATION | 0.0  |
| LOADING | 1.0  |
| 0.0      | 0.0  |

### Stresses

<table>
<thead>
<tr>
<th>PARAMETERS</th>
<th>L0 IN KIP</th>
</tr>
</thead>
<tbody>
<tr>
<td>IN KIP</td>
<td>L0 IN KIP</td>
</tr>
<tr>
<td>-------------</td>
<td>-----------</td>
</tr>
<tr>
<td>L0 FA</td>
<td>10.55</td>
</tr>
<tr>
<td>L0 fa</td>
<td>0.87</td>
</tr>
<tr>
<td>L0 FCZ</td>
<td>0.00</td>
</tr>
<tr>
<td>L0 FCY</td>
<td>0.00</td>
</tr>
<tr>
<td>L0 FTZ</td>
<td>0.00</td>
</tr>
<tr>
<td>L0 FTY</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Forces

| FZ         | 34.91     |

### Moments

| MZ         | 165.72    |

### Verification

- Absolute MZ Envelope: 68.87 KIP-FT
- Maximum Force/Moment Summary: 7.3 KIP-FT
- Stresses:
  - KL/R-Y: 28.34 KIP-FT
  - KL/R-Z: 61.76 KIP-FT
  - UNL: 93.72 KIP-FT
  - CB: 0.00 KIP-FT
  - CMY: 0.00 KIP-FT
  - CMZ: 0.00 KIP-FT
  - FYLD: 36.00 KIP-FT
  - NSF: 0.90 KIP-FT
  - DFF: 0.00 KIP-FT
### V. AISC N690 1994 Channel
Verify the design of channel section used for a beam member using the AISC N690 1994 code.

#### Details
The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

#### Validation
- **Maximum shear force (at fixed end):** \( V = 51.375 \text{ kips} \) (taken from STAAD.Pro analysis)
- **Maximum moment (at fixed end):** \( M = 830.0 \text{ in-kip} \) (taken from STAAD.Pro analysis)

#### Slenderness Ratio
- **About z-axis:** \( (KL/r)_z = 0.85(80 \text{ in})/(5.25 \text{ in}) = 12.95 \)
- **About y-axis:** \( (KL/r)_y = 0.85(80 \text{ in})/(0.865 \text{ in}) = 78.61 \)
Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

\[
\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83
\]

The laterally unsupported length of the compression flange, \( L_b = 80 < \frac{76b_f}{\sqrt{F_y}} = 120 \)

and \( \frac{20,000}{(d/A_f) F_y} = 89.56 \)

As \( b_f/t_f \) is not in the range of \( 65/\sqrt{(F_y)} \) and \( 95/\sqrt{(F_y)} \), the value of allowable bending compressive and tensile stress \( F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6 \text{ ksi} \)

Actual bending stress:

Bending Tensile Stress = bending compressive stress \( f_{btz} = f_{bcz} = MzD / (2I_z) = 15.4 \text{ ksi} \)

\[
830.0 (15.0) / (2 \times 404) = 15.41 \text{ ksi}
\]

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

\[
F_y = 51.375 \text{ kips}
\]

STAAD.Pro calculates the shear are based on the general shear stress equation, \( \tau = V \times Q / (I \times t) \) from Timoshenko. Thus the shear area, \( A_y = I \times t / Q = 8.47 \text{ in}^2 \)

\[
f_v = 51.375 / 8.47 = 6.07 \text{ ksi}
\]

Allowable shear:

\( a = \) clear distance between transverse stiffeners (here taken as length) = 80 in

\( h = \) clear distance flanges of the girder = \( D - 2t_f = 14.35 \text{ in} \)

Thus, \( a/h = 5.57 \), Therefore, \( k = 5.34 + [4.00 / (a/h)^2] = 5.46 \)

\[
t = t_w = 0.72 \text{ in}
\]

If

\[
C_v = 17.2
\]

\[
F_v = \begin{cases} 
F_y / 2.89 \times C_v = 25.6 \text{ ksi} \\
0.4 \times F_y = 14.4 \text{ ksi}
\end{cases}
\]

Hence allowable = 14.4 ksi

Therefore, Ratio = \( f_v / F_v = 6.07 / 14.4 = 0.422 \)

Results
### Table 594: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.41</td>
<td>15.41</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>6.07</td>
<td>6.06</td>
<td>negligible</td>
<td></td>
</tr>
</tbody>
</table>

#### STAAD Input

```
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1994\AISC N690 1994 Channel.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
```
## Verification Examples

### V.09 Steel Design

### S T A A D  O u t p u t

- **STAAD.PRO CODE CHECKING - ( ANSI N690-1994) v1.0**
- **PARAMETER | L1 | IN KIP INCH**
- **STRESSES | IN KIP INCH**

**96.2 (KIP-FEET)**

**KL/R/Y= 78.61 | L1 | IN KIP INCH**

**KL/R-Z= 12.97 | L1 | L1 | IN KIP INCH**

**UNL = 80.00 | L1 | L1 | IN KIP INCH**

**CB = 1.00 | L1 | L1 | IN KIP INCH**

**CMY = 1.00 | IN KIP INCH**

**CMZ = 1.00 | IN KIP INCH**

**FYLD = 36.00 | L1 | L0 | IN KIP INCH**

**NSF = 0.90 | IN KIP INCH**

**DFF = 0.00 | -3.8 | IN KIP INCH**

**(KL/R)max = 78.61 | (WITH LOAD NO.)**

**FA = 9.71 | fby = 0.00 | FV = 14.40**

**fbz = 15.41 | Fey = 21.54 | pv = 6.06**
V. AISC N690 1994 Pipe
Verify the design of a pipe section used for a beam member per the AISC N690 1994 code.

Details
The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, \( V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \)

Maximum moment, \( M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \)

Slenderness ratio = \( (KL/r) = 9.72 < 300 \)

Bending About Major Axis (Z-Axis)
Section is compact: \( \frac{OD}{t} = 36.43 < \frac{3}{300} \frac{F_y}{F} = 91.67 \)

Allowable bending stress: \( F_b = 0.66 \times F_y = 23.76 \text{ksi} \)

Actual bending stress: \( f_{bz} = f_{bcz} = \frac{M_z}{T_z} \left( \frac{OD}{2} \right) = \frac{331.25}{262} \left( \frac{12.75}{2} \right) = 8.06 \text{ksi} \)

Stress ratio: 0.339

Shear
From the reference, the shear area:
\[
A_y = \frac{0.25n(r_2^4 - r_1^4)^2(r_2 - r_1)}{2^3(r_2^3 - r_1^3)}
\]
where
\[ r_1 = \text{inner radius} \]
\[ r_2 = \text{outer radius} \]

Shear area: \( A_y = 6.8 \text{ in}^2 \)

Actual shear stress (at supports): \( f_v = \frac{25.75}{6.8} = 3.78 \text{ ksi} \)

Actual shear stress (at midspan): \( f_v = \frac{0.75}{6.8} = 0.110 \text{ ksi} \)

Allowable shear stress, \( F_v = 0.4 \times F_y = 14.4 \text{ ksi} \)

Stress ratio: \( 3.78/14.4 = 0.263 \)

Results

Table 595: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.76</td>
<td>23.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1994\AISC N690 1994 Pipe.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
```
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
MAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
STYPE 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

---

**---**

<table>
<thead>
<tr>
<th>MEMBER   1</th>
<th>AISC SECTIONS ST PIPS120</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DESIGN CODE</strong></td>
<td><strong>ANSI N690-94</strong></td>
</tr>
<tr>
<td><strong>PARAMETER</strong></td>
<td><strong>27.6 (KIP-FEET)</strong></td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th><strong>Y PROPERTIES</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>IN INCH UNIT</strong></td>
</tr>
<tr>
<td>AX = 13.70</td>
</tr>
<tr>
<td>AY = 6.80</td>
</tr>
<tr>
<td>AZ = 6.80</td>
</tr>
<tr>
<td>SY = 41.10</td>
</tr>
<tr>
<td>SZ = 41.10</td>
</tr>
<tr>
<td>RY = 4.37</td>
</tr>
<tr>
<td>RZ = 4.37</td>
</tr>
</tbody>
</table>

---

**---**
### Verification Examples

**V.09 Steel Design**

**Verification Examples**

**V. AISC N690 1994 W Shaped**
Verify the design of a wide-flange section used for a cantilever beam using the AISC N690 1994 code.

### Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

### Validation

Maximum shear force (constant shear): \( V = 12.5 \text{ kips} \)

Maximum moment (at fixed end): \( M = 12.5 \text{ kips} \times 120 \text{ in} = 1,500 \text{ in} \cdot \text{kips} \)

**Slenderness Ratio**

\((KL/r)_z = \text{Slenderness Ratio along Z-Axis} = 18.11\)

\((KL/r)_y = \text{Slenderness Ratio along Y-Axis} = 76.68\)

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

### Bending

Allowable Bending Stress About Major Axis (Z Axis)

<table>
<thead>
<tr>
<th>MAX FORCE/ MOMENT SUMMARY (KIP-FEET)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>VALUE</strong></td>
</tr>
<tr>
<td><strong>AXIAL</strong></td>
</tr>
<tr>
<td>0.00</td>
</tr>
<tr>
<td>0.00</td>
</tr>
<tr>
<td>0.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>DESIGN SUMMARY (KIP-FEET)</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>RESULT/ CRITICAL COND/ RATIO/ LOADING/</strong></td>
</tr>
<tr>
<td><strong>FX</strong></td>
</tr>
<tr>
<td>PASS</td>
</tr>
<tr>
<td>0.00 T</td>
</tr>
</tbody>
</table>

**STAAD.Pro** 4099 **User Manual**
Thus, allowable bending tensile stress \( F_{btz} = 0.66 \times F_y = 23.76 \text{ ksi} \)

**Actual bending stress About Major Axis**

\[
f_{btz} = \frac{M_z D}{2I_z} = 23.2 \text{ ksi}
\]

Stress ratio = 23.2 ksi / 23.76 ksi = 0.975

**Shear**

**Allowable Shear Stress**

\[
a = \text{clear distance between transverse stiffeners (here taken as length)} = 120 \text{ inch}
\]

\[
h = \text{clear distance flanges of the girder} = D-2t_f = 14.91 \text{ inch}
\]

Thus, \( a/h = 8.05 \), Therefore, \( k = 5.34 + \left[ \frac{4.00}{(a/h)^2} \right] = 5.36 \)

\[
t = t_w = 0.3 \text{ inch}
\]

\[
C_v = 1.475
\]

\[
F_v = \frac{F_y}{2.89} C_v \geq 0.4 \times F_y = 14.4 \text{ ksi}
\]

Hence allowable = 14.4 ksi

**Actual Shear Stress**

STAAD.Pro calculates the shear are based on the general shear stress equation, \( \tau = \frac{V \times Q}{I \times t} \) from Timoshenko. Thus the shear area, \( A_y = I \times t/Q = 4.401 \text{ in}^2 \)

\[
\tau = F_y/A_y = 12.5 \text{ kip} / 4.401 = 2.84 \text{ ksi}
\]

Stress ratio = \( \frac{\tau}{F_v} = 0.18 \)

**Results**

**Table 596: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.76</td>
<td>23.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.17</td>
<td>23.17</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.975</td>
<td>0.975</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.84</td>
<td>2.84</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1994\AISC N690 1994 W Shaped.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC N690 1994
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER 1</th>
<th>AISC SECTIONS</th>
<th>AX = 11.80</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ST W16X40</td>
<td>--Z AV = 4.39</td>
</tr>
</tbody>
</table>

**STAAD.PRO CODE CHECKING - ( ANSI N690-1994) v1.0**

*-----------------*---------*---------------*
| MEMBER 1        | AISC SECTIONS | AX = 11.80 |
| ST W16X40       | --Z AV = 4.39 |
### DESIGN SUMMARY (KIP-FEET)

<table>
<thead>
<tr>
<th>RESULT/ FX</th>
<th>CRITICAL COND/ MY</th>
<th>RATIO/ MZ</th>
<th>LOADING/ LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>PASS</td>
<td>ANSI Q1.6-2</td>
<td>0.975</td>
<td>1</td>
</tr>
<tr>
<td>0.00 T</td>
<td>0.00</td>
<td>-125.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**V.AISC N690 1984**

V. AISC N690 1984 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the AISC N690 1984 code.

**Details**

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).
Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36$ ksi, $F_u = 58$ ksi).

Assume a net section factor of 0.9, an effective length factor of 0.85, and $C_t = 0.9$.

Validation

\[ L = \sqrt{5^2 + 6^2} \text{(12in/ft)} = 93.726 \text{in} \]

Slenderness ratio (same for both members) using the minimum value of \( r \):

\[ kL/r_z = 0.85(93.726) / 1.29 = 61.76 \]

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

**Axial Tension: Member 6**

Corrected area, $A_c = C_t \times A_x = 7.57 \text{ in}^2$

Actual tensile stress $= P/A = 2,666.18 / 7.57 = 352.2 \text{ psi}$

Allowable tensile stress (Tension capacity) $= 0.6 \times F_y = 0.6 \times 36 = 21.6 \text{ ksi}$

Critical ratio $= 352.2 / 21600 = 0.016$

**Axial Compression: Member 7**

Actual compressive stress $= P/A = 7,104.23 / 8.41 = 844.74 \text{ psi}$

\[ C_c' = \sqrt{\frac{2n^2E}{Q_s Q_a F_y}} = 126.2 \]

\( kL/r \)min $< C_c'$, allowable compression stress, $F_a = 17.2 \text{ ksi}$

Critical ratio $= 844.74 / 17,200 = 0.049$

Results
### Table 597: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Member 6 (tension)</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>352</td>
<td>352</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>21,600</td>
<td>21,600</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td><strong>Member 7 (compression)</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>845</td>
<td>845</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>17,200</td>
<td>17,200</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.049</td>
<td>0.049</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1984\AISC N690 1984 Angle.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
```
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
STYPE 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

|-----------------------------------------------------------------------------
| 6 ST L806010 (AISC SECTIONS) |
| PASS Q1.5.1.1 0.016 1 |
| -2666.18 T 0.00 0.00 0.00 |
|-----------------------------------------------------------------------------

SLENDERNESS CHECK: ACTUAL RATIO: 61.76 ALLOWABLE RATIO: 300.00
ALLOWABLE STRESSES: (UNIT - POUN INCH)
AX.TENS: 2.16E+04 COMPRESS:1.72E+04

Verification Examples
V.09 Steel Design
ACTUAL STRESSES: (UNIT - POUND/INCH)
| AX.TENS: 3.52E+02 | COMPRESS: 0.00E +00 |

SECTION PROPERTIES: (UNIT - INCH)
| AXX: 8.41 AYY: 3.33 AZZ: 2.50 RZZ: 1.29 RYY: 2.81 |
| SZZ: 5.16 SYY: 11.99 |

PARAMETER: (UNIT - POUND/INCH)
| KL/R-Z: 61.76 KL/R-Y: 28.34 UNL: 60.0 CMZ: 1.00 CMY: 1.0 |
| CB: 0.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90 CT: 0.90 STEEL TYPE: 1.0 |

CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUND/INCH)
| CLAUSE       | RATIO | LOAD | FX | VY | VZ | MZ |
| TENSION      | 0.016 | 1    | 2.67E+03 | - | - | - |
| COMPRESSION  | 0.000 | 1    | 2.67E+03 | - | - | - |
| COMP&BEND    | 0.000 | 1    | 2.67E+03 | - | - | 0.00E+00 0.00E +00 |
| TEN&BEND     | 0.016 | 1    | 2.67E+03 | - | - | 0.00E+00 0.00E +00 |
| SHEAR-Y      | 0.000 | 1    | - | 0.00E+00 | - | - |
| SHEAR-Z      | 0.000 | 1    | - | - | 0.00E+00 | - |

7 ST L806010 (AISC SECTIONS)
| PASS | Q1.5.1.3.1 | 0.049 | 1 |
| 7104.23 C | 0.00 | 0.00 | 0.00 |

SLENDERNESS CHECK: ACTUAL RATIO: 61.76 ALLOWABLE RATIO: 200.00
ALLOWABLE STRESSES: (UNIT - POUND/INCH)
| AX.TENS: 2.16E+04 | COMPRESS: 1.72E +04 |
V. AISC N690 1984 Channel
Verify the design of channel section used for a beam member using the AISC N690 1984 code.

Details
The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.
The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): $V = 51.375$ kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): $M = 830.0$ in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About $z$-axis: $(KL/r)_z = 0.85(80 \text{ in})/(5.25 \text{ in}) = 12.95$

About $y$-axis: $(KL/r)_y = 0.85(80 \text{ in})/(0.865 \text{ in}) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,000}{(d / A_f)F_y} = 89.56$

As $b_f/t_f$ is not in the range of $65 / \sqrt{(F_y)}$ and $95 / \sqrt{(F_y)}$, the value of allowable bending compressive and tensile stress is $F_{bty} = F_{bco} = 0.6 \times F_y = 21.6$ ksi

Actual bending stress:

Bending Tensile Stress = bending compressive stress $= f_{btz} = f_{bcz} = MzD / (2I) = 15.4$ ksi

$$830.0 \times (15.0) / (2 \times 404) = 15.41 \text{ ksi}$$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$$F_y = 51.375 \text{ kips}$$

$$V_y = D \times t_w = 10.8 \text{ in}^2$$

$$f_{v,y} = F_y/V_y = 4.76 \text{ ksi}$$

Allowable shear:

$$a = \text{clear distance between transverse stiffeners (here taken as length)} = 80 \text{ in}$$

$$h = \text{clear distance flanges of the girder} = D-2t_f = 14.35 \text{ in}$$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/ (a/h)^2] = 5.46$

$$t = t_w = 0.72 \text{ inch}$$

$$C_v = 17.2$$

$$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4 \text{ ksi}$$

Hence allowable = 14.4 ksi

Therefore, Ratio $= f_{v}/F_v = 0.330$

Results
Table 598: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.4</td>
<td>15.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>4.76</td>
<td>4.78</td>
<td>0.4%</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1984\AISC N690 1984 Channel.std is typically installed with the program.

    START JOB INFORMATION
    ENGINEER DATE 10-Aug-18
    END JOB INFORMATION
    INPUT WIDTH 79
    SET SHEAR
    UNIT INCHES POUND
    JOINT COORDINATES
    1 0 0 0; 2 80 0 0;
    MEMBER INCIDENCES
    1 1 2;
    DEFINE MATERIAL START
    ISOTROPIC STEEL
    E 2.9e+07
    POISSON 0.3
    DENSITY 0.283
    ALPHA 1.2e-05
    DAMP 0.03
    TYPE STEEL
    STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
    END DEFINE MATERIAL
    MEMBER PROPERTY AMERICAN
    1 TABLE ST C15X50
    CONSTANTS
    MATERIAL STEEL ALL
    SUPPORTS
    1 FIXED
    2 PINNED
    LOAD 1 LOADTYPE None TITLE LOAD CASE 1
### MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80

### PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH

### STAAD Output

```
STAAD.PRO CODE CHECKING - ( AISC N690 1984) v1.0

ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST C15X50 (AISC SECTIONS)
PASS Q1.6-Eqn 2 0.713 1
0.00 T 829999.94 0.00

| SLENDERNESS CHECK: ACTUAL RATIO: 78.61 ALLOWABLE RATIO: 300.00 |
| ALLOWABLE STRESSES: (UNIT - POUN INCH) |
| AXIAL: 1.55E+04 FCZ: 2.16E+04 FCY: 2.16E+04 FTZ: 2.16E+04 FTY: 2.16E+04 |
| SHEAR: 1.44E+00 |
| ACTUAL STRESSES: (UNIT - POUN INCH) |
| AXIAL: 0.00E+00 FBZ: 1.54E+04 FBY: 0.00E+00 SHEAR: 4.78E+04 |

| SECTION PROPERTIES: (UNIT - INCH) |
| AXX: 14.70 AYY: 10.74 AZZ: 3.22 RZZ: 5.24 RYY: 0.87 |
| SZZ: 53.87 SYY: |
```

---

**STAAD.Pro**

4110

**User Manual**
Verify the design of a pipe section used for a beam member per the AISC N690 1984 code.

Reference

Details
The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, \( V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \)

Maximum moment, \( M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \)

Slenderness ratio = \( (KL/r) = 9.72 < 300 \)

Bending About Major Axis (Z-Axis)
Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76 \text{ ksi}$

Actual bending stress:

$$f_{btz} = f_{bcz} = \frac{M_z}{I_z} \left(\frac{OD}{2}\right) = \frac{331.25}{262} \left(\frac{12.75}{2}\right) = 8.06 \text{ ksi}$$

Stress ratio: 0.339

**Shear**

From the reference, the shear area:

$$A_y = \frac{0.25n(r_2^4 - r_1^4)}{2(r_2^3 - r_1^3)}$$

where

- $r_1$ = inner radius
- $r_2$ = outer radius

Shear area: $A_y = 6.8 \text{ in}^2$

Actual shear stress (at supports): $f_v = 25.75 / 6.8 = 3.78 \text{ ksi}$

Actual shear stress (at midspan): $f_v = 0.75 / 6.8 = 0.110 \text{ ksi}$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4 \text{ ksi}$

Stress ratio: $3.78 / 14.4 = 0.263$

**Results**

**Table 599: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.263</td>
<td>0.263</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

**STAAD.Pro** 4112 **User Manual**
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1984\AISC N690 1984 Pipe.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
STYPE 1 ALL
TRACK 2 ALL
### STAAD Output

**STAAD.PRO CODE CHECKING - ( AISC N690 1984) v1.0**

---

**ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)**

**MEMBER** | **RESULT/ CRITICAL COND/ RATIO/ LOADING/**
--- | --- | --- | --- | --- |
| **TABLE** | **FX** | **MY** | **MZ** | **LOCATION** |
--- | --- | --- | --- | --- |
1 | ST PIPS120 | (AISC SECTIONS) | | |
--- | --- | --- | --- | --- |
PASS | Q1.6-Eqn 2 | 0.339 | 1 |
--- | --- | --- | --- | --- |
| 0.00 T | 0.00 | -331249.97 | 25.00 |
--- | --- | --- | --- | --- |

---

**SLENDERNESS CHECK:**  
**ACTUAL RATIO:** 9.72  
**ALLOWABLE RATIO:** 300.00

---

**ALLOWABLE STRESSES:**  
(UNIT - POUN INCH)  
| AXIAL: 1.59E+04 | FCZ: 2.38E+04 | FCY: 2.38E+04 | FTZ: 2.38E+04 | FTY: 2.38E+04 |
--- | --- | --- | --- | --- |
| SHEAR: 1.44E+04 |
--- | --- | --- | --- | --- |

---

**ACTUAL STRESSES:**  
(UNIT - POUN INCH)  
| AXIAL: 0.00E+00 | FBZ: 8.06E+03 | FBY: 0.00E+00 | SHEAR: 1.10E+02 |
--- | --- | --- | --- |

---

**SECTION PROPERTIES:**  
(UNIT - INCH)  
| AXX: 13.70 | AYY: 7.01 | AZZ: 7.01 | RZZ: 4.37 | RYY: 4.37 |
--- | --- | --- | --- | --- |
| SZZ: 41.10 | SYY: 41.10 |
--- | --- | --- | --- | --- |

---

**PARAMETER:**  
(UNIT - POUN INCH)  
| KL/R-Z: 9.72 | KL/R-Y: 9.72 | UNL: 25.0 | CMZ: 1.00 | CMY: 1.00 |
--- | --- | --- | --- | --- |
| CB: 1.00 | FYLD: 36000.00 | FU: 58000.00 | NET SECTION FACTOR: 0.90 |
--- | --- | --- | --- | --- |
| CT: 0.85 | STEEL TYPE: 1.0 |
--- | --- | --- | --- | --- |

---

**CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH)**  
| CLAUSE | RATIO LOAD | FX | VY | VZ | MZ |
--- | --- | --- | --- | --- | --- |
| TENSION | 0.000 | 1 | 0.00E+00 | - | - |
--- | --- | --- | --- | --- | --- |
V. AISC N690 1984 Tee
Verify the design of a pipe section used for a beam member per the AISC N690 1984 code.

Details
A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation
Maximum shear force (at support) \( V = wL + P = (1 \text{ kip/in}) \times 60 \text{ in} + 2 \text{ kip} = 62 \text{ kips} \)

Maximum moment (at support) \( M = 0.5 \times w \times L^2 + P \times L = 0.5 (1 \text{ kip/in}) \times (60 \text{ in})^2 + (2 \text{ kip}) \times (60 \text{ in}) = 1,920 \text{ in kips} \)

Slenderness Ratio
Slenderness Ratio along Z-Axis \( (KL/r)_z = 21.22 \)
Slenderness Ratio along Y-Axis \( (KL/r)_y = 50.53 \)
Maximum Slenderness Ratio \( 50.53 < 300 \)

Bending
Allowable Bending Stress About Major Axis (Z Axis)
\[
b \frac{d}{2t} = 4.65 > 65/\sqrt{(F_Y)} = 10.833 \\
d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67
\]

Thus allowable bending stress is obtained by treating the section as compact.
\( F_b = 0.66 \times F_Y = 23.8 \text{ ksi} \)

Actual Bending Stress
Bending Tensile Stress \( f_{bz} = (Mz/Iz) \times Y_{bar} = 14.93 \text{ ksi} < 23.8 \text{ ksi} \)
Bending Compressive Stress \( f_{bcz} = 38.7 \text{ ksi} > 23.8 \text{ ksi} \)
Critical stress ratio \( = 38.7 \text{ ksi} / 23.8 \text{ ksi} = 1.626 \)

Shear
Allowable Shear Stress
\( a = \text{clear distance between transverse stiffeners (here taken as length)} = 60 \text{ in} \)
h = clear distance flanges of the girder = D-t = 17.01 in
Thus, a/h = 3.527 Therefore, \( k = 5.34 + \left[ \frac{4.00}{(a/h)^2} \right] = 5.65 \)
\[ t = t_w = 0.44 \text{ inch} \]
If \( C_v > 0.8 \), \( C_v = \left[ \frac{190}{(h/t)} \right] \sqrt{\frac{k}{S_y}} = 78.4 \) which is true. Thus, the value of \( C_v \) is taken as 78.4
Allowable shear stress = \( F_v = \left( \frac{F_y}{2.89} \right) C_v \leq 0.4F_y = 14.4 \text{ ksi} \)
Hence allowable shear =14.4 ksi
Actual Shear
\[ V_y = D t_w = 10.68 \text{ in}^2 \]
\[ f_v = \frac{F_y}{V_y} = 5.805 \text{ ksi} \]
Stress ratio = \( f_v/F_y = 0.403 \)
Allowable bending stress about minor axis, \( f_{btz} = 0.75 \times F_y = 27 \text{ ksi} \)
Results

<table>
<thead>
<tr>
<th>Table 600: Verification comparison for AISC_N690_1984_Tee</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Parameter</strong></td>
</tr>
<tr>
<td>slenderness ratio</td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
</tr>
<tr>
<td>Critical shear ratio</td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
</tr>
<tr>
<td>Shear stress ratio</td>
</tr>
</tbody>
</table>

STAAD Input
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1984\AISC N690 1984 Tee.std is typically installed with the program.
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
STYPE 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - ( AISC N690 1984) v1.0
********************************************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
                  FX MY MZ LOCATION
=======================================================================
*    1  T    W36X135                  (AISC SECTIONS)
FAIL     Q1.6-Eqn 2        1.620         1
0.00 T          0.00     1919999.75        0.00
SLENDERNESS CHECK: ACTUAL RATIO: 50.53 ALLOWABLE RATIO: 300.00
ALLOWABLE STRESSES: (UNIT - POUND/INCH)
  | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.70E+04  FTZ: 2.38E+04  FTY: 2.70E+04
  | SHEAR: 1.44E+04
ACTUAL STRESSES: (UNIT - POUND/INCH)
  | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00  SHEAR: 5.81E+03
SECTION PROPERTIES: (UNIT - INCH)
  | AXX: 19.95  AYY: 9.61  AZZ: 6.32  RZZ: 5.66  RYY: 2.37  SZZ: 49.69  SYY: 18.75
PARAMETER: (UNIT - POUND/INCH)
  | KL/R-Z: 21.22  KL/R-Y: 50.53  UNL: 60.0  CMZ: 1.00  CMY: 1.00  CB: 1.00  FYLD: 36000.00  FU: 58000.00  NET SECTION FACTOR: 0.90  CT: 0.85  STEEL TYPE: 0.0
CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUND/INCH)
  | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ MY
  | TENSION      0.000    1 0.00E+00       -         -         -
  | COMPRESSION 0.000    1 0.00E+00       -         -         -
  | COMP&BEND   1.620    1 0.00E+00       -         - 1.92E+06 0.00E+00
  | TEN&BEND    0.000    1 0.00E+00       -         - 1.92E+06 0.00E+00
  | SHEAR-Y     0.403    1 - 6.20E+04       -         -
  | SHEAR-Z     0.000    1 - - 0.00E+00       -
V. AISC N690 1984 W Shaped
Verify the design of a wide-flange section used for a cantilever beam using the AISC N690 1984 code.

Details
The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.
The member is a W16X40 section with Grade A36 steel.

Validation
Maximum shear force (constant shear): \( V = 12.5 \text{ kips} \)
Maximum moment (at fixed end): \( M = 12.5 \text{ kips} \times 120 \text{ in} = 1500 \text{ in} \cdot \text{kips} \)

Slenderness Ratio
\[(KL/r)_{z} = \text{Slenderness Ratio along Z-Axis} = 18.11\]
\[(KL/r)_{y} = \text{Slenderness Ratio along Y-Axis} = 76.68\]
Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

Bending
Allowable Bending Stress About Major Axis (Z Axis)
\[\phi_{b}2t_{f} = 6.862 < 65/\sqrt{F_{y}}\]
Thus, allowable bending tensile stress \( F_{btz} = 0.66 \times F_{y} = 23.76 \text{ ksi} \)
Actual bending stress About Major Axis
\[f_{btz} = M_{z}D / 2 \times I_{z} = 23.2 \text{ ksi}\]
Stress ratio = 23.2 ksi / 23.76 ksi = 0.975

Shear
Allowable Shear Stress
\[F_{v} = 0.4 \times F_{y} = 14.4 \text{ ksi}\]
Actual Shear Stress
\[V_{y} = D \times t_{w} = 4.8 \text{ in}^{2}\]
\[f_{v} = F_{y}/V_{y} = 2.6 \text{ ksi}\]
Stress ratio = \( f_{v}/F_{v} = 0.18\)

Results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.2</td>
<td>23.2</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

Table 601: Comparison of results
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical stress ratio</td>
<td>0.975</td>
<td>0.975</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.60</td>
<td>2.56</td>
<td>1.5%</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.180</td>
<td>0.178</td>
<td>1.1%</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\N690\1984\AISC N690 1984 W Shaped.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC N690 1984
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
```
### STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1 ST</td>
<td>W16X40</td>
<td>PASS</td>
<td>Q1.6-Eqn 2</td>
<td>0.975</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
<td>-1500.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**SLENDERNESS CHECK:**
- **ACTUAL RATIO:** 76.68
- **ALLOWABLE RATIO:** 240.00

**ALLOWABLE STRESSES:**
- **AXIAL:** 1.49E+01 FCZ: 2.38E+01 FCY: 2.70E+01 FTZ: 2.38E+01 FTY: 2.70E+01
- **SHEAR:** 1.44E+01

**ACTUAL STRESSES:**
- **AXIAL:** 0.00E+00 FBZ: 2.32E+01 FBY: 0.00E+00 SHEAR: 2.56E+01

**SECTION PROPERTIES:**
- **AXX:** 11.80
- **AYY:** 4.88
- **AZZ:** 4.71
- **RZZ:** 6.63
- **RYY:** 1.56
- **SZZ:** 64.75
- **SYY:** 8.26

**PARAMETER:**
- **KL/R-Z:** 18.11
- **KL/R-Y:** 76.68
- **UNL:** 72.0
- **CMZ:** 0.85
- **CMY:** 0.85
- **CB:** 1.00
- **FYLD:** 36.00
- **FU:** 58.02
- **NET SECTION FACTOR:** 1.00
- **CT:** 0.75
- **STEEL TYPE:** 0.0

**CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP - INCH):**
- **CLAUSE**
- **RATIO**
- **LOAD**
- **FX**
- **VY**
- **VZ**
- **MZ**
V. ASME NF 3000 Codes

V. NF 3000-1974

V. ASME NF 3000 1974 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1974 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).
Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36$ ksi, $F_u = 58$ ksi).
Assume a net section factor of 0.9, an effective length factor of 0.85, and $C_t = 0.9$.

Validation

\[ L = \sqrt{5^2 + 6^2} \text{in/ft} = 93.726 \text{in} \]

Slenderness ratio (same for both members) using the minimum value of \( r \):

\[ kL/r_z = 0.85(93.726)/1.29 = 61.76 \]

For member 6, \( kL/r_z = 61.76 < 300 \)

For member 7, \( kL/r_z = 61.76 < 200 \)

**Axial Tension: Member 6 (per NF-74.2211)**

Corrected area, \( A_c = C_t \times A_x = 7.57 \text{ in}^2 \)

Actual tensile stress \( = P/A = 2,666.18 / 7.57 = 352.2 \text{ psi} \)

Allowable tensile stress (Tension capacity) \( = 0.6 \times F_y = 0.6 \times 36 = 21.6 \text{ ksi} \)

Critical ratio \( = 352.2/21,600 = 0.016 \)

**Axial Compression: Member 7 (NF-74.2213):**

Actual compressive stress \( = P/A = 7,104.23 / 8.41 = 844.74 \text{ psi} \)

\[ C_c' = \sqrt{\frac{2n^2E}{Q_s Q_a F_y}} = 126.2 \quad \text{(NF-3322.2(e)(5))} \]

\( (kL/r)_{min} < C_c', \) allowable compression stress, \( F_a = 17.2 \text{ ksi} \)

Critical ratio \( = 844.74 / 17,200 = 0.049 \)

Results
Table 602: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member 6 (tension)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>352</td>
<td>352</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>21,600</td>
<td>21,600</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Member 7 (compression)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>845</td>
<td>845</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>17,200</td>
<td>17,200</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.049</td>
<td>0.049</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1974\ASME NF 3000 1974 Angle.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
### Verification Examples

**V.09 Steel Design**

```plaintext
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1974
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

#### STAAD Output

<table>
<thead>
<tr>
<th></th>
<th>PASS</th>
<th>NF-74.2211</th>
<th>0.016</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>-2668.18 T</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>SLENDEINESS CHECK: ACTUAL RATIO: 72.65 ALLOWABLE RATIO:</th>
</tr>
</thead>
<tbody>
<tr>
<td>300.00</td>
<td>ALLOWABLE STRESSES: (UNIT - POUN INCH)</td>
</tr>
<tr>
<td></td>
<td>AX.TENS: 2.16E+04 COMPRESS:1.72E</td>
</tr>
<tr>
<td>+04</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>ACTUAL STRESSES: (UNIT - POUN INCH)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>AX.TENS: 3.52E+02 COMPRESS:0.00E</td>
</tr>
<tr>
<td>+00</td>
<td></td>
</tr>
</tbody>
</table>
```

**STAAD.Pro**

4125

*User Manual*
### SECTION PROPERTIES: (UNIT - INCH)
- AXX: 8.41
- AYY: 3.33
- AZZ: 2.50
- RZZ: 1.29
- RYY: 2.81
- SZZ: 5.16
- SYY: 11.99

### PARAMETER: (UNIT - POUND INCH)
- KL/R-Z: 61.76
- KL/R-Y: 28.34
- UNL: 60.0
- CMZ: 1.00
- CMY: 1.00
- CB: 0.00
- FYLD: 36000.00
- FU: 58000.00
- NET SECTION FACTOR: 0.90
- KS: 1.000
- KV: 1.000
- KBK: 1.000

### CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUND- INCH)

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.016</td>
<td>1</td>
<td>2.67E+03</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
<td>0.016</td>
<td>1</td>
<td>2.67E+03</td>
<td>-</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>SHEAR-Y</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>

### SLENDERNESS CHECK
- ACTUAL RATIO: 61.76
- ALLOWABLE RATIO: 200.00
- ALLOWABLE STRESSES: (UNIT - POUND INCH)
  - AX TENS: 2.16E+04
  - COMPRESS: 1.72E+04

### ACTUAL STRESSES: (UNIT - POUND INCH)
V. ASME NF 3000 1974 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 1974 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.
Validation

Maximum shear force (at fixed end): \( V = 51.375 \) kips (taken from STAAD.Pro analysis)
Maximum moment (at fixed end): \( M = 830.0 \) in-kip (taken from STAAD.Pro analysis)

**Slenderness Ratio**

About \( z \)-axis: \( \frac{(KL/r)_{z}}{z} = 0.85(80 \text{ in})/(5.25 \text{ in}) = 12.95 \)
About \( y \)-axis: \( \frac{(KL/r)_{y}}{y} = 0.85(80 \text{ in})/(0.865 \text{ in}) = 78.61 \)
Max. slenderness ratio < 300, OK.

**Bending About Major Axis (Z Axis)**

\[
\frac{b_{f}}{t_{f}} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83
\]

The laterally unsupported length of the compression flange, \( L_{b} = 80 \) < \( \frac{76b_{f}}{\sqrt{F_y}} = 120 \)

and \( < \frac{20,000}{(d/A_f)F_y} = 89.56 \)

As \( b_{f}/t_{f} \) is not in the range of \( 65/\sqrt{(F_y)} \) and \( 95/\sqrt{(F_y)} \), the value of allowable bending compressive and tensile stress \( F_{by} = F_{bcy} = 0.6\times F_y = 21.6 \text{ ksi} \)

Actual bending stress:

**Bending Tensile Stress** = bending compressive stress = \( f_{btz} = f_{bcz} = \frac{MzD}{(2I_{z})} = 15.4 \text{ ksi} \)

\[
830.0 \text{ (15.0) / (2 \times 404)} = 15.41 \text{ ksi}
\]

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

**Shear**

Actual shear:

\[
F_y = 51.375 \text{ kips}
\]

\[
V_y = D \times t_w = 10.8 \text{ in}^2
\]

\[
f_{v,y} = \frac{F_y}{V_y} = 4.76 \text{ ksi}
\]

Allowable shear:

\( a = \) clear distance between transverse stiffeners (here taken as length) = 80 in
\( h = \) clear distance flanges of the girder = \( D-2t_f = 14.35 \text{ in} \)

Thus, \( a/h = 5.57, \) Therefore, \( k = 5.34 + \left[ \frac{4.00}{(a/h)^2} \right] = 5.46 \)

\[
t = t_w = 0.72 \text{ inch}
\]

\[
C_v = 17.2
\]

\[
F_y = (F_y/2.89)C_v \geq 0.4\times F_y = 14.4 \text{ ksi}
\]

Hence allowable=14.4 ksi

Therefore, Ratio = \( f_v/F_y = 0.330 \)

Results
Table 603: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.4</td>
<td>15.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>4.76</td>
<td>4.78</td>
<td>0.4%</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1974\ASME NF 3000 1974 Channel.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1974
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - (ASME NF3000-74) v2.0

******************************************************************************

ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)

MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================

1 ST C15X50 (AISC SECTIONS)
PASS NF-74-EQN-21 0.713 1
0.00 T 0.00 829999.94 0.00
|----------------------------------------------------------------------------|
| SLENDERNESS CHECK: ACTUAL RATIO: 78.61 ALLOWABLE RATIO:
300.00 |
<p>| ALLOWABLE STRESSES: (UNIT - POUN INCH) |
| AXIAL: 1.55E+04 FCZ: 2.16E+04 FCY: 2.16E+04 FTZ: 2.16E+04 FTY: 2.16E+04 |
| +04 |
| SHEAR: 1.44E +04 |
| ACTUAL STRESSES: (UNIT - POUN INCH) |</p>
<table>
<thead>
<tr>
<th>AXIAL: 0.00E+00 FBZ: 1.54E+04 FBY: 0.00E+00 SHEAR: 4.78E +03</th>
</tr>
</thead>
<tbody>
<tr>
<td>SECTION PROPERTIES: (UNIT - INCH)</td>
</tr>
<tr>
<td>AXX: 14.70 AYY: 10.74 AZZ: 3.22 RZZ: 5.24 RYY: 0.87</td>
</tr>
<tr>
<td>SZZ: 53.87 SYY:</td>
</tr>
</tbody>
</table>
V. ASME NF 3000 1974 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1974 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, \( V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \)

Maximum moment, \( M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \)

Slenderness ratio = \( \frac{KL}{r} = 9.72 < 300 \)

Bending About Major Axis (Z-Axis)
Section is compact: \( \frac{OD}{t} = 36.43 < \frac{3 \times 300}{F_y} = 91.67 \)

Allowable bending stress: \( F_b = 0.66 \times F_y = 23.76 \text{ ksi} \)

Actual bending stress: \( f_{btz} = f_{bcz} = \frac{M_z}{T_z} \left( \frac{OD}{2} \right) = \frac{331.25}{262} \left( \frac{12.75}{2} \right) = 8.06 \text{ ksi} \)

Stress ratio: 0.339

Shear

From the reference, the shear area:

\[
A_y = \frac{0.25n(r_2^4 - r_1^4)2(r_2 - r_1)}{2(r_2^3 - r_1^3)}
\]

where

\[
r_1 = \text{inner radius} \\
r_2 = \text{outer radius}
\]

Shear area: \( A_y = 6.8 \text{ in}^2 \)

Actual shear stress (at supports): \( f_v = 25.75 / 6.8 = 3.78 \text{ ksi} \)

Actual shear stress (at midspan): \( f_v = 0.75 / 6.8 = 0.110 \text{ ksi} \)

Allowable shear stress, \( F_v = 0.4 \times F_y = 14.4 \text{ ksi} \)

Stress ratio: 3.78/14.4 = 0.263

Results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.201</td>
<td>0.201</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1974\ASME NF 3000 1974 Pipe.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1974
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
### STAAD Output

**STAAD.PRO CODE CHECKING - ( ASME NF3000-74) v2.0**  
*ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)*

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
</tbody>
</table>

| 1    | ST          | PIPS120   | PASS           | NF-74-EQN-21 | 0.339     | 1         |
|      |             |           | 0.00 T         | 0.00     | -331249.97 | 25.00    |

**SLENDERNESS CHECK:**  
ACTUAL RATIO: 9.72  ALLOWABLE RATIO: 300.00  
ALLOWABLE STRESSES: (UNIT - POUN INCH)  
| AXIAL: 2.12E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E+04 |
| SHEAR: 1.44E+04 |

**ACTUAL STRESSES:** (UNIT - POUN INCH)  
| AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00  SHEAR: 8.43E+01  |

**SECTION PROPERTIES:** (UNIT - INCH)  

**PARAMETER:** (UNIT - POUN INCH)  
| KL/R-Z: 9.72  KL/R-Y: 9.72  UNL: 25.0  CMZ: 1.00  CMY: 1.00  CB: 1.00  FYLD: 36000.00  FU: 58000.00  NET SECTION FACTOR: 0.90  KS: 1.000  KV: 1.000  KBK: 1.000  |

**CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH):**  
<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
<td>0.339</td>
<td>1</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>3.31E+05 0.00E</td>
</tr>
</tbody>
</table>
Verify the design of a pipe section used for a beam member per the ASME NF 3000 1974 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = \( V = wL + P = (1 \text{ kip/in}) \times 60 \text{ in} + 2 \text{ kip} = 62 \text{ kips} \)

Maximum moment (at support) = \( M = 0.5 \times w \times L^2 + P \times L = 0.5 (1 \text{ kip/in}) \times (60 \text{ in})^2 + (2 \text{ kip}) \times (60 \text{ in}) = 1,920 \text{ in·kips} \)

Slenderness Ratio

Slenderness Ratio along Z-Axis = \( (KL/r)_z = 21.22 \)

Slenderness Ratio along Y-Axis = \( (KL/r)_y = 50.53 \)

Maximum Slenderness Ratio = 50.53 < 300

Bending

Allowable Bending Stress About Major Axis (Z Axis)
\[
b_f/2t_z = 4.65 > 65/\sqrt{(F_Y)} = 10.833
\]
\[
d/w = 28.3 < 640/\sqrt{(F_Y)} = 106.67
\]

Thus allowable bending stress is obtained by treating the section as compact.

\( F_b = 0.66 \times F_Y = 23.8 \text{ ksi} \)

Actual Bending Stress

Bending Tensile Stress = \( f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93 \text{ ksi} < 23.8 \text{ ksi} \)

Bending Compressive Stress = \( f_{bcz} = 38.7 \text{ ksi} > 23.8 \text{ ksi} \)

Critical stress ratio = 38.7 ksi / 23.8 ksi = 1.626

Shear

Allowable Shear Stress

\( a = \text{clear distance between transverse stiffeners (here taken as length)} = 60 \text{ in} \)

\( h = \text{clear distance flanges of the girder} = D-t_f = 17.01 \text{ in} \)
Thus, $a/h = 3.527$ Therefore, $k = 5.34 + \frac{4.00}{(a/h)^2} = 5.65$

$t = t_w = 0.44$ inch

If $Cv > 0.8$, $Cv = \frac{190}{(h/t)} \sqrt{\frac{k}{Sy}} = 78.4$ which is true. Thus, the value of $Cv$ is taken as 78.4

Allowable shear stress $= F_v = (FY/2.89)Cv \leq 0.4F_y = 14.4$ ksi

Hence allowable shear $= 14.4$ ksi

Actual Shear

$V_y = D t_w = 10.68$ in$^2$

$f_v = F_y/V_y = 5.805$ ksi

Stress ratio $= f_v/F_y = 0.403$

Results

Table 605: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>50.53</td>
<td>50.53</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>38.7</td>
<td>38.5</td>
<td>0.5%</td>
<td></td>
</tr>
<tr>
<td>Critical shear ratio</td>
<td>1.626</td>
<td>1.620</td>
<td>0.4%</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>5.81</td>
<td>5.81</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.403</td>
<td>0.403</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1974\ASME NF 3000 1974 Tee.std is typically installed with the program.
STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-74) v2.0
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
   FX MY MZ LOCATION
=======================================================================
*    1 T W36X135   (AISC SECTIONS)  FAIL NF-74-EQN-21  1.620  1
   0.00 T 0.00 1919999.75 0.00
| SLENDERNESS CHECK: ACTUAL RATIO: 50.53 ALLOWABLE RATIO:
300.00 |
### Allowable Stresses:

<table>
<thead>
<tr>
<th>Stress Type</th>
<th>Allowable Value (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>1.16E+04</td>
</tr>
<tr>
<td>Shear</td>
<td>1.44E+04</td>
</tr>
<tr>
<td>Shear-FCZ</td>
<td>2.38E+04</td>
</tr>
<tr>
<td>Shear-FCY</td>
<td>2.16E+04</td>
</tr>
<tr>
<td>Shear-FTZ</td>
<td>2.16E+04</td>
</tr>
<tr>
<td>Shear-FTY</td>
<td>2.16E+04</td>
</tr>
</tbody>
</table>

### Actual Stresses:

<table>
<thead>
<tr>
<th>Stress Type</th>
<th>Actual Value (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Shear</td>
<td>5.81E+03</td>
</tr>
<tr>
<td>Shear-FBZ</td>
<td>3.85E+04</td>
</tr>
<tr>
<td>Shear-FBY</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

### Section Properties:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>AXX</td>
<td>19.95</td>
</tr>
<tr>
<td>AYY</td>
<td>9.61</td>
</tr>
<tr>
<td>AZZ</td>
<td>6.32</td>
</tr>
<tr>
<td>RZZ</td>
<td>5.66</td>
</tr>
<tr>
<td>RYY</td>
<td>2.37</td>
</tr>
<tr>
<td>SZZ</td>
<td>49.69</td>
</tr>
<tr>
<td>SYY</td>
<td>18.75</td>
</tr>
</tbody>
</table>

### Parameter:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Z</td>
<td>21.22</td>
</tr>
<tr>
<td>KL/R-Y</td>
<td>50.53</td>
</tr>
<tr>
<td>UNL</td>
<td>60.0</td>
</tr>
<tr>
<td>CMZ</td>
<td>1.00</td>
</tr>
<tr>
<td>CMY</td>
<td>1.00</td>
</tr>
<tr>
<td>CB</td>
<td>1.00</td>
</tr>
<tr>
<td>FYLD</td>
<td>36000.00</td>
</tr>
<tr>
<td>FU</td>
<td>58000.00</td>
</tr>
<tr>
<td>KS</td>
<td>1.000</td>
</tr>
<tr>
<td>KV</td>
<td>1.000</td>
</tr>
</tbody>
</table>

### Critical Loads for Each Clause Check:

<table>
<thead>
<tr>
<th>Clause</th>
<th>Ratio</th>
<th>Load</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tension</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Compression</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Comp&amp;Bend</td>
<td>1.620</td>
<td>1</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>1.92E+06 0.00E</td>
</tr>
<tr>
<td>Ten&amp;Bend</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>0.00E+00 0.00E</td>
</tr>
<tr>
<td>Shear-Y</td>
<td>0.403</td>
<td>1</td>
<td>-</td>
<td>6.20E+04</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Shear-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

---

**V. ASME NF 3000 1974 WShaped**

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1974 code.
Details
The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.
The member is a W16X40 section with Grade A36 steel.

Validation
Maximum shear force (constant shear): \( V = 12.5 \text{ kips} \)
Maximum moment (at fixed end): \( M = 12.5 \text{ kips} \times 120 \text{ in} = 1,500 \text{ in \cdot kips} \)

Slenderness Ratio
\( (KL/r)_z = \text{Slenderness Ratio along Z-Axis} = 18.11 \)
\( (KL/r)_y = \text{Slenderness Ratio along Y-Axis} = 76.68 \)
Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

Bending about Major Axis (Z Axis)
Allowable Bending Stress
\[ b_f/2t_f < 6.862 < 65/(F_y)^{0.5} \] [Ref. NF3322.1(d)(2)]
Thus, allowable bending tensile stress = \( f_{btz} = F_y \times [0.79 - 0.002 \times (b_f/2t_f) \times \sqrt{F_y}] = 25.47 \text{ ksi} \)

Actual Bending Stress
Bending Tensile Stress = Bending Compressive Stress = \( f_{btz} = f_{bcz} = M_z D/2I_z = 23.17 \text{ ksi} \)
Stress ratio = 23.16 ksi / 25.47 ksi = 0.909

Shear

Allowable Shear Stress
\( a = \text{clear distance between transverse stiffeners (here taken as length)} = 120 \text{ in} \)
\( h = \text{clear distance flanges of the girder} = D - 2t_f = 14.91 \text{ in} \)
Thus, \( a/h = 8.05 \), Therefore, \( k = 5.34 + [4.00/(a/h)^2] = 5.36 \) [Ref. NF3322.6(e)(2)]
\( t = t_w = 0.3 \text{ inch} \)
\( Cv = 1.475 \)
Allowable shear stress = \( F_v = (F_y/2.89)Cv = > 0.4F_y = 14.4 \text{ ksi} \) Hence allowable=14.4ksi

Actual Shear Stress
\( F_y = 12.5 \text{ kip, Vy} = D t_w = 4.8 \text{ inch}^2 \)
\( f_v = F_y/V_y = 2.6 \text{ ksi} \)
Stress ratio = \( f_v/F_v = 0.18 \)

Results

Table 606: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
### Parameter

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>25.47</td>
<td>24.3</td>
<td>4.8%</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.2</td>
<td>23.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.909</td>
<td>0.954</td>
<td>4.6%</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.6</td>
<td>2.56</td>
<td>1.5%</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.18</td>
<td>0.178</td>
<td>1.1%</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1974\ASME NF 3000 1974 W Shaped.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Mar-19
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
2 FY 12.5
PERFORM ANALYSIS
```
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 1974
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-74) v2.0
******************************************************************************
ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST W16X40 (AISC SECTIONS)
PASS NF-74-EQN-21 0.954 1
0.00 T 0.00 -1500.00 0.00
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SLENDERNESS CHECK: ACTUAL RATIO: 76.68 ALLOWABLE RATIO:</td>
</tr>
</tbody>
</table>
240.00 |
| ALLOWABLE STRESSES: (UNIT - KIP |
INCH) |
| AXIAL: 1.51E+01 FCZ: 2.43E+01 FCY: 2.70E+01 FTZ: 2.43E+01 FTY: 2.70E+01 |
| SHEAR: 1.44E+01 |
| ACTUAL STRESSES: (UNIT - KIP |
INCH) |
<table>
<thead>
<tr>
<th>AXIAL: 0.00E+00 FBZ: 2.32E+01 FBY: 0.00E+00 SHEAR: 2.56E+00</th>
</tr>
</thead>
<tbody>
<tr>
<td>SECTION PROPERTIES: (UNIT -</td>
</tr>
</tbody>
</table>
INCH) |
<table>
<thead>
<tr>
<th>SZZ: 64.75 SYY: 8.26</th>
</tr>
</thead>
<tbody>
<tr>
<td>PARAMETER: (UNIT - KIP</td>
</tr>
</tbody>
</table>
INCH) |
| KL/R-Z: 18.11 KL/R-Y: 76.68 UNL: 72.0 CMZ: 0.85 CMY: 0.85 |
| CB: 1.00 FYLD: 36.00 FU: 58.02 NET SECTION FACTOR: 1.00 |
| KS: 1.000 KV: 1.000 KBK: 1.000 |
| CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP - INCH) |
| CLAUSE       | RATIO | LOAD | FX  | VY  | VZ  | MZ  |
| MY   |
| TENSION      | 0.000 | 0    | 0.00E+00 | -   | -   | -   |
| -   |
| COMPRESSION  | 0.000 | 0    | 0.00E+00 | -   | -   | -   |
| -   |
| COMP&BEND    | 0.954 | 1    | 0.00E+00 | -   | -   | 1.50E+03 0.00E |
| +00 |
| TEN&BEND     | 0.000 | 0    | 0.00E+00 | -   | -   | 0.00E+00 0.00E |
| +00 |
| SHEAR-Y      | 0.178 | 1    | -    | 1.25E+01 | -   | -   |
| -   |
| SHEAR-Z      | 0.000 | 0    | -    | -   | 0.00E+00 | -   |
| -   |

V. NF 3000-1977

V. ASME NF 3000 1977 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1977 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are "truss" members (axial-only).
Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36 \text{ ksi}, F_u = 58 \text{ ksi}$).
Assume a net section factor of 0.9, an effective length factor of 0.85, and $C_t = 0.9$.

Validation

$$ L = \sqrt{5^2 + 6^2} \times (12 \text{ in/ft}) = 93.726 \text{ in} $$

Slenderness ratio (same for both members) using the minimum value of $r$: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$
For member 7, $kL/r_z = 61.76 < 200$

**Axial Tension: Member 6 (per NF-77.2211)**

Corrected area, $A_c = C_t \times A_x = 7.57 \text{ in}^2$

Actual tensile stress $= P/A = 2,666.18 \div 7.57 = 352.2 \text{ psi}$

Allowable tensile stress (Tension capacity) $= 0.6 \times F_y = 0.6 \times 36 = 21.6 \text{ ksi}$

Critical ratio $= 352.2/21,600 = 0.016$

**Axial Compression: Member 7 (NF-77.2213):**

Actual compressive stress $= P/A = 7,104.23 \div 8.41 = 844.74 \text{ psi}$

$$ C_c' = \sqrt{\frac{2n^2 E}{Q_e Q_a F_y}} = 126.2 \hspace{1cm} \text{(NF-3322.2(e)(5))} $$

$(kL/r)_{min} < C_c'$, allowable compression stress $F_a = 17.2 \text{ ksi}$

Critical ratio $= 844.74 / 17,200 = 0.049$

Results
Table 607: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member 6 (tension)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>352</td>
<td>352</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>21,600</td>
<td>21,600</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Member 7 (compression)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>845</td>
<td>845</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>17,200</td>
<td>17,200</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.049</td>
<td>0.049</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1977\ASME NF 3000 1977 Angle.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8x48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output
6 ST L806010 (AISC SECTIONS)
PASS NF-77.2211 0.016 1
-2666.18 T 0.00 0.00 0.00

| SLENDERNESS CHECK: ACTUAL RATIO: 72.65 ALLOWABLE RATIO: 300.00 |
| ALLOWABLE STRESSES: (UNIT - POUn INCH) |
| AX.TENS: 2.16E+04 COMPRESS: 1.72E |
| +04 |
| ACTUAL STRESSES: (UNIT - POUn INCH) |
| AX.TENS: 3.52E+02 COMPRESS: 0.00E |
| +00 |
SECTION PROPERTIES: (UNIT - INCH)
| AXX: 8.41 AYY: 3.33 AZZ: 2.50 RZZ: 1.29 RYY: 2.81 |
| SZZ: 5.16 SYY: 11.99 |

PARAMETER: (UNIT - POUND-INCH)
| KL/R-Z: 61.76 KL/R-Y: 28.34 UNL: 60.0 CMZ: 1.00 CMY: 1.00 |
| CB: 0.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90 |
| KS:1.000 KV:1.000 KBK: 1.000 |

CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUND-INCH)
| CLAUSE | RATIO | LOAD | FX | VY | VZ | MZ |
| TENSION | 0.016 | 1 | 2.67E+03 | - | - | - |
| COMPRESSION | 0.000 | 0 | 0.00E+00 | - | - | - |
| COMP&BEND | 0.000 | 0 | 0.00E+00 | - | - | 0.00E+00 0.00E+00 |
| TEN&BEND | 0.016 | 1 | 2.67E+03 | - | - | 0.00E+00 0.00E+00 |
| SHEAR-Y | 0.000 | 0 | - | 0.00E+00 | - | - |
| SHEAR-Z | 0.000 | 0 | - | - | 0.00E+00 | - |

STL806010 (AISC SECTIONS)
| PASS | NF-77.2213 | 0.849 | 1 |
| 7104.23 C | 0.00 | 0.00 | 0.00 |

SLENDERNESS CHECK: ACTUAL RATIO: 61.76 ALLOWABLE RATIO: 200.00

ALLOWABLE STRESSES: (UNIT - POUND-INCH)
| AX.TENS: 2.16E+04 | COMPRESS: 1.72E+04 |

ACTUAL STRESSES: (UNIT - POUND)
V. ASME NF 3000 1977 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 1977 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.
Validation
Maximum shear force (at fixed end): \( V = 51.375 \text{ kips} \) (taken from STAAD.Pro analysis)
Maximum moment (at fixed end): \( M = 830.0 \text{ in-kip} \) (taken from STAAD.Pro analysis)

Slenderness Ratio
About z-axis: \((KL/r)_z = 0.85(80 \text{ in})/(5.25 \text{ in}) = 12.95\)
About y-axis: \((KL/r)_y = 0.85(80 \text{ in})/(0.865 \text{ in}) = 78.61\)
Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)
\[
\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83
\]
The laterally unsupported length of the compression flange, \( L_b = 80 < \frac{76b_f}{\sqrt{F_y}} = 120\)

As \(b_f/t_f\) is not in the range of \(65/\sqrt{(Fy)}\) and \(95/\sqrt{(Fy)}\), the value of allowable bending compressive and tensile stress = \(F_{by} = F_{bcy} = 0.6\times F_y = 21.6 \text{ ksi}\)

Actual bending stress:
Bending Tensile Stress = bending compressive stress = \(f_{btz} = f_{bcz} = MzD / (2I_z) = 15.4 \text{ ksi}\)

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear
Actual shear:
\[
F_y = 51.375 \text{ kips}
\]
\[
V_y = D\times t_w = 10.8 \text{ in}^2
\]
\[
f_{v,y} = F_y/V_y = 4.76 \text{ksi}
\]
Allowable shear:
an = clear distance between transverse stiffeners (here taken as length) = 80 in
h = clear distance flanges of the girder = \(D-2t_f = 14.35 \text{ in}\)
Thus, \(a/h = 5.57\), Therefore, \(k = 5.34 + [4.00/ (a/h)^2] = 5.46\)
\[
t = t_w = 0.72 \text{ inch}
\]
\[
Cv = 17.2
\]
\[
F_v = (F_y/2.89)C_v \geq 0.4\times F_y = 14.4 \text{ksi}
\]
Hence allowable=14.4 ksi
Therefore, Ratio = \(f_v/F_v = 0.330\)

Results
### Table 608: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.4</td>
<td>15.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>4.76</td>
<td>4.78</td>
<td>0.4%</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1977\ASME NF 3000 1977 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
```
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-77) v2.0
*****************************************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST C15X50 (AISC SECTIONS) NF-77-EQN-21 0.713 1 0.00 T 0.00 829999.94 0.00
----------------------------------------------------------------------------
| SLENDERNESS CHECK: ACTUAL RATIO: 78.61 ALLOWABLE RATIO: 300.00 |
| ALLOWABLE STRESSES: (UNIT - POUN INCH) AXIAL: 1.55E+04 FCZ: 2.16E+04 FCY: 2.16E+04 FTZ: 2.16E+04 FTY: 2.16E+04 |
| SHEAR: 1.44E+04 |
| ACTUAL STRESSES: (UNIT - POUN INCH) AXIAL: 0.00E+00 FBZ: 1.54E+04 FBY: 0.00E+00 SHEAR: 4.78E+03 |
----------------------------------------------------------------------------
| SECTION PROPERTIES: (UNIT - INCH) AXX: 14.70 AYY: 10.74 AZZ: 3.22 RZZ: 5.24 RYY: 0.87 |
| SZZ: 53.87 SYY: |
### ASME NF 3000 1977 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1977 code.

**Details**

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

**Validation**

Maximum shear force, \( V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \)

Maximum moment, \( M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \)

Slenderness ratio = \( \frac{KL}{r} = 9.72 < 300 \)

**Bending About Major Axis (Z-Axis)**
Section is compact: $\frac{OD}{t} = 36.43 < \frac{3}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76$ ksi

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z} \left(\frac{OD}{2}\right) = \frac{331.25}{262} \left(\frac{12.75}{2}\right) = 8.06$ ksi

Stress ratio: 0.339

Shear

From the reference, the shear area:

$$A_y = \frac{0.25n(r_2^4 - r_1^4)2(r_2 \cdot r_1)}{2(r_2^3 - r_1^3)}$$

where

$r_1$ = inner radius
$r_2$ = outer radius

Shear area: $A_y = 6.8$ in$^2$

Actual shear stress (at supports): $f_v = 25.75 / 6.8 = 3.78$ ksi

Actual shear stress (at midspan): $f_v = 0.75 / 6.8 = 0.110$ ksi

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4$ ksi

Stress ratio: $3.78/14.4 = 0.263$

Results

Table 609: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress at mid-span (ksi)</td>
<td>0.084</td>
<td>0.0843</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio at supports</td>
<td>0.201</td>
<td>0.201</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1977\ASME NF 3000 1977 Pipe.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
### STAAD Output

STAAD.PRO CODE CHECKING - (ASME NF3000-77) v2.0  
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%  
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)  
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/  
FX MY MZ LOCATION  
=======================================================================

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST</td>
<td>PIPS120</td>
<td>PASS NF-77-EQN-21 0.339 1 0.00 T 0.00 -331249.97 25.00</td>
</tr>
</tbody>
</table>

| SLENDERNESS CHECK: ACTUAL RATIO: 9.72 ALLOWABLE RATIO: 300.00 |
| ALLOWABLE STRESSES: (UNIT - POUN INCH) |
| AXIAL: 2.12E+04 FCZ: 2.38E+04 FCY: 2.38E+04 FTZ: 2.38E+04 FTY: 2.38E+04 |
| SHEAR: 1.44E04 |

| ACTUAL STRESSES: (UNIT - POUN INCH) |
| AXIAL: 0.00E+00 FBZ: 8.06E+03 FBY: 0.00E+00 SHEAR: 8.43E+01 |

| SECTION PROPERTIES: (UNIT - INCH) |
| AXX: 13.70 AYY: 7.01 AZZ: 7.01 RZZ: 4.37 RYY: 4.37 |
| SZZ: 41.10 SYY: 41.10 |

| PARAMETER: (UNIT - POUN INCH) |
| KL/R-Z: 9.72 KL/R-Y: 9.72 UNL: 25.0 CMZ: 1.00 CMY: 1.00 |
| CB: 1.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90 |
| KS:1.000 KV:1.000 KBK: 1.000 |

| CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH) |
| CLAUSE RATIO LOAD FX VY VZ MZ |
| TENSION 0.000 0 0.00E+00 - - - |
| COMPRESSION 0.000 0 0.00E+00 - - - |
| COMP&BEND 0.339 1 0.00E+00 - - 3.31E+05 0.00E |
Verify the design of a pipe section used for a beam member per the ASME NF 3000 1977 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = \( V = wL + P = (1 \text{kip/in}) \times 60 \text{in} + 2 \text{kip} = 62 \text{kips} \)

Maximum moment (at support) = \( M = 0.5 \times w \times L^2 + P \times L = 0.5 \times (1 \text{kip/in}) \times (60 \text{in})^2 + (2 \text{kip}) \times (60 \text{in}) = 1,920 \text{in kips} \)

Slenderness Ratio

Slenderness Ratio along Z-Axis = \( (KL/r)_z = 21.22 \)

Slenderness Ratio along Y-Axis = \( (KL/r)_y = 50.53 \)

Maximum Slenderness Ratio = 50.53 < 300

Bending

Allowable Bending Stress About Major Axis (Z Axis)

\[
\frac{b_f^2t_f}{2t_f} = 4.65 > 65/\sqrt{(F_Y)} = 10.833
\]

\[
d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67
\]

Thus allowable bending stress is obtained by treating the section as compact.

\[
F_b = 0.66 \times F_Y = 23.8 \text{ksi}
\]

Actual Bending Stress

Bending Tensile Stress = \( f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93 \text{ksi} < 23.8 \text{ksi} \)

Bending Compressive Stress = \( f_{bcz} = 38.7 \text{ksi} > 23.8 \text{ksi} \)

Critical stress ratio = \( 38.7 \text{ksi} / 23.8 \text{ksi} = 1.626 \)

Shear

Allowable Shear Stress

\[
a = \text{clear distance between transverse stiffeners (here taken as length)} = 60 \text{in}
\]

\[
h = \text{clear distance flanges of the girder} = D - t_f = 17.01 \text{in}
\]
Thus, \( a/h = 3.527 \) Therefore, \( k = 5.34 + \left[ \frac{4.00}{(a/h)^2} \right] = 5.65 \)

\[ t = t_w = 0.44 \text{ inch} \]

If \( Cv > 0.8 \), \( Cv = \left[ \frac{190}{(h/t)} \right] \sqrt{(k/Sy)} = 78.4 \) which is true. Thus, the value of \( Cv \) is taken as 78.4

Allowable shear stress \( = F_v = (FY/2.89)Cv \leq 0.4F_y = 14.4 \text{ ksi} \)

Hence allowable shear \( = 14.4 \text{ ksi} \)

Actual Shear

\[ V_y = D t_w = 10.68 \text{ in}^2 \]

\[ f_v = F_y/V_y = 5.805 \text{ ksi} \]

Stress ratio \( = f_v/F_v = 0.403 \)

Results

Table 610: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>50.53</td>
<td>50.53</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>38.7</td>
<td>38.5</td>
<td>0.5%</td>
<td></td>
</tr>
<tr>
<td>Critical shear ratio</td>
<td>1.626</td>
<td>1.620</td>
<td>0.4%</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>5.81</td>
<td>5.81</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.403</td>
<td>0.403</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

```
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\V09 Steel Design\US\ASME\NF 3000-1977\ASME NF 3000 1977 Tee.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
```
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL

1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-77) v2.0
******************************************************************************
ALL UNITS ARE - PO UN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
   FX   MY   MZ LOCATION
******************************************************************************
* 1 T W36X135 (AISC SECTIONS)
   FAIL NF-77-EQN-21  1.620   1
   0.00 T  0.00  1919999.75  0.00

| SLENDERNESS CHECK: ACTUAL RATIO: 50.53 ALLOWABLE RATIO:
300.00 |
### ALLOWABLE STRESSES: (UNIT - POUN INCH)
- **AXIAL**: 1.16E+04
- **FCZ**: 2.38E+04
- **FCY**: 2.16E+04
- **FTZ**: 2.16E+04
- **FTY**: 2.16E+04
- **SHEAR**: 1.44E+04

### ACTUAL STRESSES: (UNIT - POUN INCH)
- **AXIAL**: 0.00E+00
- **FBZ**: 3.85E+04
- **FBY**: 0.00E+00
- **SHEAR**: 5.81E+03

### SECTION PROPERTIES: (UNIT - INCH)
- **AXX**: 19.95
- **AYY**: 9.61
- **AZZ**: 6.32
- **RZZ**: 5.66
- **RYY**: 2.37
- **SZZ**: 49.69
- **SYY**: 18.75

### PARAMETER: (UNIT - POUN INCH)
- **KL/R-Z**: 21.22
- **KL/R-Y**: 50.53
- **UNL**: 60.0
- **CMZ**: 1.00
- **CMY**: 1.00
- **CB**: 1.00
- **FYLD**: 36000.00
- **FU**: 58000.00
- **NET SECTION FACTOR**: 0.90
- **KS**: 1.000
- **KV**: 1.000
- **KB**: 1.000

### CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH)
<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
<td>1.620</td>
<td>1</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>1.92E+06 0.00E +00</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>0.00E+00 0.00E +00</td>
</tr>
<tr>
<td>SHEAR-Y</td>
<td>0.403</td>
<td>1</td>
<td>-</td>
<td>6.20E+04</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>

---

**V. ASME NF 3000 1977 WS shaped**

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1977 code.
**Details**

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

**Validation**

Maximum shear force (constant shear): \( V = 12.5 \text{ kips} \)

Maximum moment (at fixed end): \( M = 12.5 \text{ kips} \times 120 \text{ in} = 1,500 \text{ in} \cdot \text{kips} \)

**Slenderness Ratio**

\( (KL/r)_z = \text{Slenderness Ratio along Z-Axis} = 18.11 \)

\( (KL/r)_y = \text{Slenderness Ratio along Y-Axis} = 76.68 \)

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

**Bending about Major Axis [Z Axis]**

Allowable Bending Stress

\[ b_f/2t_f = 6.862 < 65/(F_y)^{0.5} \] [Ref. NF3322.1(d)(2)]

Thus, allowable bending tensile stress = \( F_{btz} = F_Y \times (0.79 - 0.002 \times (b_f/2t_f)) \times \sqrt{F_y} = 25.47 \text{ ksi} \)

**Actual Bending Stress**

Bending Tensile Stress = Bending Compressive Stress = \( f_{btz} = f_{bcz} = M_z D / 2I_z = 23.17 \text{ ksi} \)

Stress ratio = 23.16 ksi / 25.47 ksi = 0.909

**Shear**

Allowable Shear Stress

\[ a = \text{clear distance between transverse stiffeners (here taken as length)} = 120 \text{ in} \]

\[ h = \text{clear distance flanges of the girder} = D - 2t_f = 14.91 \text{ in} \]

Thus, \( a/h = 8.05 \), Therefore, \( k = 5.34 + [4.00 / (a/h)^2] = 5.36 \) [Ref. NF3322.6(e)(2)]

\[ t = t_w = 0.3 \text{ inch} \]

\[ Cv = 1.475 \]

Allowable shear stress = \( F_v = (F_y/2.89)Cv = > 0.4F_y = 14.4 \text{ ksi} \) Hence allowable=14.4ksi

**Actual Shear Stress**

\[ F_y = 12.5 \text{ kip, } V_y = D t_w = 4.8 \text{ inch}^2 \]

\[ f_v = F_y/V_y = 2.6 \text{ ksi} \]

Stress ratio = \( f_v/F_v = 0.18 \)

**Results**

**Table 611: Verification comparison for ASME_NF_3000_1977_WShaped**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Parameter</td>
<td>Hand Calculation</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>------------------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>25.47</td>
<td>24.3</td>
<td>4.8%</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.2</td>
<td>23.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.909</td>
<td>0.954</td>
<td>4.6%</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.6</td>
<td>2.56</td>
<td>1.5%</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.18</td>
<td>0.178</td>
<td>1.1%</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1977\ASME NF 3000 1977 W Shaped.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Mar-19
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
2 FY 12.5
PERFORM ANALYSIS
## Staad Output

**Staad.Pro Code Checking - (ASME NF3000-77) v2.0**

*All units are - KIP inch (unless otherwise noted)*

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>ST</td>
<td>W16X40</td>
<td>PASS</td>
<td>NF-77-EQN-21</td>
<td>0.954</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
<td>-1500.00</td>
</tr>
</tbody>
</table>

**SLENDERNESS CHECK:**  
Actual Ratio: 76.68  Allowable Ratio: 240.00

**Allowable Stresses:** (Unit - KIP inch)  
- Axial: 1.51E+01  
- CZ: 2.43E+01  
- Cy: 2.70E+01  
- Fz: 2.43E+01  
- Fy: 2.70E+01

**Actual Stresses:** (Unit - KIP inch)  
- Axial: 0.00E+00
- CZ: 2.32E+01  
- Cy: 0.00E+00

**Section Properties:** (Unit - inch)  
- AXX: 11.80  
- AYY: 4.88  
- AZZ: 4.71  
- RZZ: 6.63  
- RYY: 1.56  
- SZZ: 64.75  
- SYY: 8.26

| Parameter: (Unit - KIP inch)  
| KL/R-Z: 18.11  
| KL/R-Y: 76.68  
| UNL: 72.0  
| CMZ: 0.85  
| CMY: 0.85  
| CB: 1.00  
| FYLD: 36.00  
| FU: 58.02  
| Net Section Factor: 1.00  
| KS: 1.000  
| KV: 1.000  
| KBK: 1.000
<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
<td>0.954</td>
<td>1</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>1.50E+03 0.00E</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>0.00E+00 0.00E</td>
</tr>
<tr>
<td>SHEAR-Y</td>
<td>0.178</td>
<td>1</td>
<td>-</td>
<td>1.25E+01</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>

V. NF 3000-1989

V. ASME NF 3000 1989 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1989 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).
Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36$ ksi, $F_u = 58$ ksi).

Assume a net section factor of 0.9, an effective length factor of 0.85, and $C_t = 0.9$.

**Validation**

$$L = \sqrt{5^2 + 6^2} (12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of $r$: $kL/r = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r = 61.76 < 300$

For member 7, $kL/r = 61.76 < 200$

**Axial Tension: Member 6 (per NF-89.1.a)**

Corrected area, $A_c = C_t \times A_x = 7.57 \text{in}^2$

Actual tensile stress = $P/A = 2666.18 / 7.57 = 352.2 \text{psi}$

Allowable tensile stress (Tension capacity) = $0.6 \times F_y = 0.6 \times 36 = 21.6 \text{ksi}$

Critical ratio = $352.2/21,600 = 0.016$

**Axial Compression: Member 7 (NF-89.1.c):**

Actual compressive stress = $P/A = 7,104.23 / 8.41 = 844.74 \text{psi}$

$$C'_C = \sqrt{\frac{2n}{Q_s Q_u F_y}} = 126.2 \text{ (NF-3322.2(e)(5) )}$$

$(kL/r)_{min} < C'_C$, allowable compression stress, $F_a = 17.2 \text{ksi}$

Critical ratio = $844.74 / 17,200 = 0.049$
### Table 612: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member 6 (tension)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>352</td>
<td>352</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>21,600</td>
<td>21,600</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Member 7 (compression)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>845</td>
<td>845</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>17,200</td>
<td>17,200</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.049</td>
<td>0.049</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1989\ASME NF 3000 1989 Angle.std is typically installed with the program.

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>START JOB INFORMATION</td>
</tr>
<tr>
<td>ENGINEER DATE 26-Aug-18</td>
</tr>
<tr>
<td>END JOB INFORMATION</td>
</tr>
<tr>
<td>SET SHEAR</td>
</tr>
<tr>
<td>INPUT WIDTH 79</td>
</tr>
<tr>
<td>UNIT INCHES POUND</td>
</tr>
<tr>
<td>JOINT COORDINATES</td>
</tr>
<tr>
<td>1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;</td>
</tr>
<tr>
<td>MEMBER INCIDENCES</td>
</tr>
<tr>
<td>1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;</td>
</tr>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td>E 2.9e+07</td>
</tr>
<tr>
<td>POISSON 0.3</td>
</tr>
<tr>
<td>DENSITY 0.283</td>
</tr>
<tr>
<td>ALPHA 1.2e-05</td>
</tr>
<tr>
<td>DAMP 0.03</td>
</tr>
<tr>
<td>TYPE STEEL</td>
</tr>
<tr>
<td>STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2</td>
</tr>
<tr>
<td>END DEFINE MATERIAL</td>
</tr>
<tr>
<td>MEMBER PROPERTY AMERICAN</td>
</tr>
<tr>
<td>1 TO 5 TABLE ST W8X48</td>
</tr>
</tbody>
</table>
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1989
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>6  ST L806010 (AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PASS NF-89.1.a 0.016 1</td>
</tr>
<tr>
<td>-2666.18 T 0.00 0.00 0.00</td>
</tr>
</tbody>
</table>

| SLENDERNESS CHECK: ACTUAL RATIO: 72.65 ALLOWABLE RATIO: |
| 300.00 |
| ALLOWABLE STRESSES: (UNIT - POUND)
| AX.TENS: 2.16E+04 COMPRESS:1.72E |
| +04 |
| |
| ACTUAL STRESSES: (UNIT - POUND)
| AX.TENS: 3.52E+02 COMPRESS:0.00E |
| +00 |
SECTION PROPERTIES: (UNIT - INCH)
| AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:  2.81 |
| SZZ:   5.16  SYY:  11.99 |

PARAMETER: (UNIT - POUND INCH)
| KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:  1.00 |
| CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR: 0.90 |
| CT: 0.90   STEEL TYPE: 0.0  KS: 1.000  KV: 1.000  KBK: 1.000 |

CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUND-INCH)
| CLAUSE  | RATIO  | LOAD    | FX     | VY     | VZ     | MZ   |
| TENSION | 0.016  | 1       | 2.67E+03 | -      | -      | -   |
| COMPRESSION | 0.000 | 0 | 0.00E+00 | -      | -      | -   |
| COMP&BEND | 0.000 | 0 | 0.00E+00 | -      | 0.00E+00 | 0.00E+00 |
| TEN&BEND | 0.016 | 1 | 2.67E+03 | -      | 0.00E+00 | 0.00E+00 |
| SHEAR-Y | 0.000 | 0 | - | 0.00E+00 | -      | -   |
| SHEAR-Z | 0.000 | 0 | - | - | 0.00E+00 | -   |

ST L806010 (AISC SECTIONS)
| PASS | NF-89.1.c | 0.049 |
| 7104.23 C | 0.00 | 0.00 | 0.00 |

SLENDERNESS CHECK: ACTUAL RATIO: 61.76 ALLOWABLE RATIO: 200.00

ALLOWABLE STRESSES: (UNIT - POUND INCH)
| AX.TENS: 2.16E+04  | COMPRESS: 1.72E+04 |

ACTUAL STRESSES: (UNIT - POUND)
V. ASME NF 3000 1989 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 1989 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.
Validation

Maximum shear force (at fixed end): \( V = 51.375 \text{ kips (taken from STAAD.Pro analysis)} \)

Maximum moment (at fixed end): \( M = 830.0 \text{ in-kip (taken from STAAD.Pro analysis)} \)

Slenderness Ratio

About z-axis: \( (KL/r)_z = 0.85(80 \text{ in})/(5.25 \text{ in}) = 12.95 \)

About y-axis: \( (KL/r)_y = 0.85(80 \text{ in})/(0.865 \text{ in}) = 78.61 \)

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

\[
\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83
\]

The laterally unsupported length of the compression flange, \( L_b = 80 < \frac{76b_f}{\sqrt{F_y}} = 120 \)

and \( < \frac{20,000}{(d/A_f)F_y} = 89.56 \)

As \( b_f/t_f \) is not in the range of \( 65/\sqrt{(F_y)} \) and \( 95/\sqrt{(F_y)} \), the value of allowable bending compressive and tensile stress \( F_{by} = F_{bcy} = 0.6 \times F_y = 21.6 \text{ ksi} \)

Actual bending stress:

Bending Tensile Stress = bending compressive stress = \( f_{btz} = f_{bcz} = MzD / (2I_z) = 15.4 \text{ ksi} \)

830.0 \( (15.0) / (2 \times 404) = 15.41 \text{ ksi} \)

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

\( F_y = 51.375 \text{ kips} \)

\( V_y = D \times t_w = 10.8 \text{ in}^2 \)

\( f_{v,y} = F_y/V_y = 4.76 \text{ ksi} \)

Allowable shear:

\( a = \text{clear distance between transverse stiffeners (here taken as length)} = 80 \text{ in} \)

\( h = \text{clear distance flanges of the girder} = D-2t_f = 14.35 \text{ in} \)

Thus, \( a/h = 5.57, \) Therefore, \( k = 5.34 + [4.00/ (a/h)^2] = 5.46 \)

\( t = t_w = 0.72 \text{ inch} \)

\( C_v = 17.2 \)

\( F_y = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4 \text{ ksi} \)

Hence allowable=14.4 ksi

Therefore, Ratio = \( f_v/F_v = 0.330 \)

Results
## Table 613: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.4</td>
<td>15.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>4.76</td>
<td>4.78</td>
<td>0.4%</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1989\ASME NF 3000 1989 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
```
STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-89) v2.0
****************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER    TABLE    RESULT/   CRITICAL COND/     RATIO/       LOADING/
           FX       MY       MZ       LOCATION
=======================================================================
 1 ST       C15X50     PASS     NF-89-Eqn-22  0.713       1
          0.00 T     0.00      829999.94  0.00
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SLENDERNESS CHECK:    ACTUAL RATIO: 78.61 ALLOWABLE RATIO: 300.00</td>
</tr>
<tr>
<td>ALLOWABLE STRESSES: (UNIT - POUN INCH)</td>
</tr>
<tr>
<td>AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E+04</td>
</tr>
<tr>
<td>SHEAR: 1.44E+04</td>
</tr>
<tr>
<td>----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>ACTUAL STRESSES: (UNIT - POUN INCH)</td>
</tr>
<tr>
<td>AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00  SHEAR: 4.78E+03</td>
</tr>
<tr>
<td>----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>SECTION PROPERTIES: (UNIT - INCH)</td>
</tr>
<tr>
<td>AXX: 14.70  AYY: 10.74  AZZ: 3.22  RZZ: 5.24  RYY: 0.87</td>
</tr>
<tr>
<td>SZZ: 53.87  SYY:</td>
</tr>
</tbody>
</table>
V. ASME NF 3000 1989 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1989 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, \[ V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \]

Maximum moment, \[ M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)^2}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \]

Slenderness ratio = \( \frac{KL}{r} \) = 9.72 < 300

Bending About Major Axis (Z-Axis)
Section is compact: \[ \frac{OD}{t} = 36.43 < \frac{3.300}{F_y} = 91.67 \]

Allowable bending stress: \[ F_b = 0.66 \times F_y = 23.76 \text{ ksi} \]

Actual bending stress: \[ f_{btz} = f_{bcz} = \frac{M_z}{I_z} \left( \frac{OD}{2} \right) = \frac{331.25}{262} \left( \frac{12.75}{2} \right) = 8.06 \text{ ksi} \]

Stress ratio: 0.339

**Shear**

From the reference, the shear area:

\[ A_y = \frac{0.25n(r_2^4 - r_1^4)2(r_2 \cdot r_1)}{2(r_2^3 - r_1^3)} \]

where

- \( r_1 = \) inner radius
- \( r_2 = \) outer radius

Shear area: \( A_y = 6.8 \text{ in}^2 \)

Actual shear stress (at supports): \( f_v = 25.75 / 6.8 = 3.78 \text{ ksi} \)

Actual shear stress (at midspan): \( f_v = 0.75 / 6.8 = 0.110 \text{ ksi} \)

Allowable shear stress, \( F_v = 0.4 \times F_y = 14.4 \text{ ksi} \)

Stress ratio: \( 3.78/14.4 = 0.263 \)

**Results**

**Table 614: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.201</td>
<td>0.201</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

**Verification Examples**

V.09 Steel Design

STAAD.Pro 4172 User Manual
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1989\ASME NF 3000 1989 Pipe.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1989
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
### STAAD Output

**STAAD.PRO CODE CHECKING - (ASME NF3000-89) v2.0**

********************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION
=======================================================================
1 ST PIPS120 (AISC SECTIONS)
PASS NF-89-Eqn-22 0.339 1
0.00 T 0.00 -331249.97 25.00

| SLENDERNESS CHECK: ACTUAL RATIO: 9.72 ALLOWABLE RATIO: 300.00 |
| ALLOWABLE STRESSES: (UNIT - POUN INCH) |
| AXIAL: 2.12E+04 FCZ: 2.38E+04 FCY: 2.38E+04 FTZ: 2.38E+04 FTY: 2.38E+04 |
| SHEAR: 1.44E+04 |
| ACTUAL STRESSES: (UNIT - POUN INCH) |
| AXIAL: 0.00E+00 FBZ: 8.06E+03 FBY: 0.00E+00 SHEAR: 8.43E+01 |

| SECTION PROPERTIES: (UNIT - INCH) |
| AXX: 13.70 AYY: 7.01 AZZ: 7.01 RZZ: 4.37 RYY: 4.37 |
| SZZ: 41.10 SYY: 41.10 |

| PARAMETER: (UNIT - POUN INCH) |
| KL/R-Z: 9.72 KL/R-Y: 9.72 UNL: 25.0 CMZ: 1.00 CMY: 1.00 |
| CB: 1.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90 |
| CT: 0.85 STEEL TYPE: 0.0 KS:1.000 KV:1.000 KBK: 1.000 |

| CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH) |
| CLAUSE RATIO LOAD FX VY VZ MZ |
| TENSION 0.000 0 0.00E+00 - - |
| COMPRESSION 0.000 0 0.00E+00 - - |
| COMP&BEND 0.339 1 0.00E+00 - - 3.31E+05 0.00E |
Verify the design of a pipe section used for a beam member per the ASME NF 3000 1989 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) \( V = wL + P = (1 \text{ kip/in}) \times 60 \text{ in} + 2 \text{ kip} = 62 \text{ kips} \)

Maximum moment (at support) \( M = 0.5 \times w \times L^2 + P \times L = 0.5 (1 \text{ kip/in}) \times (60 \text{ in})^2 + (2 \text{ kip}) \times (60 \text{ in}) = 1,920 \text{ in.kips} \)

Slenderness Ratio

Slenderness Ratio along Z-Axis \( (KL/r)_z = 21.22 \)

Slenderness Ratio along Y-Axis \( (KL/r)_y = 50.53 \)

Maximum Slenderness Ratio \( = 50.53 < 300 \)

Bending

Allowable Bending Stress About Major Axis (Z Axis)
\[
\frac{b}{2t_f} = 4.65 > 65/\sqrt{F_Y} = 10.833
\]
\[
d/w = 28.3 < 640/\sqrt{F_Y} = 106.67
\]

Thus allowable bending stress is obtained by treating the section as compact.

\( F_b = 0.66 \times F_Y = 23.8 \text{ ksi} \)

Actual Bending Stress

Bending Tensile Stress \( f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93 \text{ ksi} < 23.8 \text{ ksi} \)

Bending Compressive Stress \( f_{bcz} = 38.7 \text{ ksi} > 23.8 \text{ ksi} \)

Critical stress ratio \( = 38.7 \text{ ksi} / 23.8 \text{ ksi} = 1.626 \)

Shear

Allowable Shear Stress
\[
a = \text{ clear distance between transverse stiffeners (here taken as length)} = 60 \text{ in}
\]
\[
h = \text{ clear distance flanges of the girder} = D-t_f = 17.01 \text{ in}
\]
Thus, $a/h = 3.527$ Therefore, $k = 5.34 + \frac{4.00}{(a/h)^2} = 5.65$

$t = t_w = 0.44$ inch

If $Cv > 0.8$, $Cv = \frac{190}{(h/t)} \sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of $Cv$ is taken as 78.4.

Allowable shear stress $= F_v = (FY/2.89)Cv \leq 0.4FY = 14.4$ ksi

Hence allowable shear $= 14.4$ ksi.

Actual Shear

$V_y = Dt_w = 10.68$ in$^2$

$f_v = F_y/V_y = 5.805$ ksi

Stress ratio $= f_v/F_v = 0.403$

Results

Table 615: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>50.53</td>
<td>50.53</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>38.7</td>
<td>38.5</td>
<td>0.5%</td>
<td></td>
</tr>
<tr>
<td>Critical shear ratio</td>
<td>1.626</td>
<td>1.620</td>
<td>0.4%</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>5.81</td>
<td>5.81</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.403</td>
<td>0.403</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1989\ASME NF 3000 1989 Tee.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1989
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX  MY  MZ  LOCATION</td>
</tr>
</tbody>
</table>
| *  1   | T W36X135 | FAIL NF-89-Eqn-22 1.620  1
|       |          | 0.00 T 0.00 1919999.75 0.00         |
|       |          | SLENDERNESS CHECK: ACTUAL RATIO: 50.53 ALLOWABLE RATIO: 300.00 |
Verification Examples

V. ASME NF 3000 1989 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1989 code.
Details
The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.
The member is a W16X40 section with Grade A36 steel.
Validation
Maximum shear force (constant shear): \( V = 12.5 \text{ kips} \)
Maximum moment (at fixed end): \( M = 12.5 \text{ kips} \times 120 \text{ in} = 1,500 \text{ in} \cdot \text{kips} \)
Slenderness Ratio
\( (KL/r)_z = \text{Slenderness Ratio along Z-Axis} = 18.11 \)
\( (KL/r)_y = \text{Slenderness Ratio along Y-Axis} = 76.68 \)
Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)
Bending about Major Axis [Z Axis]
Allowable Bending Stress
\[ \frac{b_f/2t_f}{2} < 65/(F_y)^{0.5} \text{ [Ref. NF3322.1(d)(2)]} \]
Thus, allowable bending tensile stress = \( F_{btz} = F_y \times [0.79-0.002 \times (b_f/2t_f) \times \sqrt{F_y}] = 25.47 \text{ ksi} \)
Actual Bending Stress
\[ \text{Bending Tensile Stress} = \text{Bending Compressive Stress} = f_{btz} = f_{bcz} = M_z D / 2I_z = 23.17 \text{ ksi} \]
Stress ratio = 23.16 ksi / 25.47 ksi = 0.909
Shear
\[ a = \text{clear distance between transverse stiffeners (here taken as length)} = 120 \text{ in} \]
\[ h = \text{clear distance flanges of the girder} = D - 2t_f = 14.91 \text{ in} \]
Thus, \( a/h = 8.05 \), Therefore, \( k = 5.34 + [4.00 / (a/h)^2] = 5.36 \text{ [Ref. NF3322.6(e)(2)]} \)
\[ t = t_w = 0.3 \text{ inch} \]
\[ C_v = 1.475 \]
Allowable shear stress = \( F_v = (F_y/2.89)C_v = > 0.4F_y = 14.4 \text{ ksi} \) Hence allowable=14.4ksi
Actual Shear Stress
\[ F_y = 12.5 \text{ kip}, \ V_y = D t_w = 4.8 \text{ inch}^2 \]
\[ f_v = F_y/V_y = 2.6 \text{ ksi} \]
Stress ratio = \( f_v/F_v = 0.18 \)
Results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Parameter</td>
<td>Hand Calculation</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>----------------------------------------</td>
<td>------------------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>25.47</td>
<td>25.4</td>
<td>0.3%</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.2</td>
<td>23.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.909</td>
<td>0.910</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.6</td>
<td>2.56</td>
<td>1.5%</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.18</td>
<td>0.178</td>
<td>1.1%</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1989\ASME NF 3000 1989 W Shaped.std is typically installed with the program.

**STAAD SPACE**

START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 1989
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>1</td>
<td>ST</td>
<td>W16X40</td>
<td>PASS</td>
<td>NF-89-Eqn-22</td>
<td>0.910</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
<td>-1500.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

SLENDERNESS CHECK: ACTUAL RATIO: 76.68 ALLOWABLE RATIO: 240.00
ALLOWABLE STRESSES: (UNIT - KIP INCH)
| AXIAL: 1.49E+01 | FCZ: 2.54E+01 | FCY: 2.70E+01 | FTZ: 2.54E+01 | FTY: 2.70E+01 |
| SHEAR: 1.44E+01 |

ACTUAL STRESSES: (UNIT - KIP INCH)
| AXIAL: 0.00E+00 | FBZ: 2.32E+01 | FBY: 0.00E+00 | SHEAR: 2.56E+00 |

SECTION PROPERTIES: (UNIT - INCH)
| SZZ: 64.75 | SYY: 8.26 |

PARAMETER: (UNIT - KIP INCH)
| KL/R-Z: 18.11 | KL/R-Y: 76.68 | UNL: 72.0 | CMZ: 0.85 | CMY: 0.85 |
| CB: 1.00 | FYLD: 36.00 | FU: 58.02 | NET SECTION FACTOR: 1.00 |
| CT: 0.75 | STEEL TYPE: 0.0 | KS: 1.000 | KV: 1.000 | KBK: |
V. NF 3000-1998

V. ASME NF 3000 1998 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1998 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).
Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36 \text{ ksi}$, $F_{u} = 58 \text{ ksi}$).
Assume a net section factor of 0.9, an effective length factor of 0.85, and $C_t = 0.9$.

Validation

$$L = \sqrt{5^2 + 6^2} \ (12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of $r$: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$
For member 7, $kL/r_z = 61.76 < 200$

Axial Tension: Member 6 (per NF-98.1.a)
Corrected area, $A_c = C_t \times A_x = 7.57 \text{ in}^2$
Actual tensile stress $= P/A = 2,666.18 / 7.57 = 352.2 \text{ psi}$
Allowable tensile stress (Tension capacity) $= 0.6 \times F_y = 0.6 \times 36 = 21.6 \text{ ksi}$
Critical ratio $= 352.2/21,600 = 0.016$

Axial Compression: Member 7 (NF-98.1.c):  
Actual compressive stress $= P/A = 7,104.23 / 8.41 = 844.74 \text{ psi}$

$$C_c' = \sqrt{\frac{2n^2E}{Q_x Q_y F_y}} = 126.2 \quad \text{(NF-3322.2(e)(5))}$$

$(kL/r)_{min} < C_c'$, allowable compression stress, $F_{a} = 17.2 \text{ ksi}$
Critical ratio $= 844.74 / 17,200 = 0.049$

Results
### Table 617: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Member 6 (tension)</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>352</td>
<td>352</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>21,600</td>
<td>21,600</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td><strong>Member 7 (compression)</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>845</td>
<td>845</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>17,200</td>
<td>17,200</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.049</td>
<td>0.049</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1998\ASME NF 3000 1998 Angle.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
```
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1998
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 30000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

| SLENDERNESS CHECK: ACTUAL RATIO: 72.65 ALLOWABLE RATIO: |
| ALLOWABLE STRESSES: (UNIT - POUND INCH) |
| AX.TENS: 2.16E+04 COMPRESS:1.72E |
| +04 | |
| ACTUAL STRESSES: (UNIT - POUND INCH) |
| AX.TENS: 3.52E+02 COMPRESS:0.00E |
| +00 | |
SECTION PROPERTIES: (UNIT - INCH)
| AXX:  8.41  AYY:  3.33  AZZ:  2.50  RZZ:  1.29  RYY: 2.81 |
| SZZ:  5.16  SYY: 11.99 |

PARAMETER: (UNIT - POUND-INCH)
| KL/R-Z: 61.76  KL/R-Y: 28.34  UNL: 60.0  CMZ: 1.00  CMY: 1.00 |
| CB: 0.00  FYLD: 36000.00  FU: 58000.00  NET SECTION FACTOR: 0.90 |
| CT: 0.90  STEEL TYPE: 0.0  KS:1.000  KV:1.000  KBK: 1.000 |

CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS - POUND-INCH)
| CLAUSE       RATIO  LOAD       FX  VY  VZ  MZ MY |
| TENSION      0.016  1  2.67E+03  -  -  - - |
| COMPRESSION  0.000  0  0.00E+00  -  -  - - |
| COMP&BEND    0.000  0  0.00E+00  -  -  0.00E+00 0.00E+00 |
| TEN&BEND     0.016  1  2.67E+03  -  -  0.00E+00 0.00E+00 |
| SHEAR-Y      0.000  0  -  0.00E+00  -  -  - |
| SHEAR-Z      0.000  0  -  -  0.00E+00  -  - |

ST L806010 (AISC SECTIONS)
| PASS  NF-98.1.C  0.049  1 |
| 7104.23 C  0.00  0.00  0.00 |

SLENDERNESS CHECK: ACTUAL RATIO: 61.76 ALLOWABLE RATIO: 200.00
ALLOWABLE STRESSES: (UNIT - POUND-INCH)
| AX.TENS: 2.16E+04  COMPRESS:1.72E+04 |

ACTUAL STRESSES: (UNIT - POUND-INCH)
### Verification Examples

#### V.09 Steel Design

<table>
<thead>
<tr>
<th>SECTION PROPERTIES: (UNIT - INCH)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>AX: 8.41 AY: 3.33 AZ: 2.50 RZ: 1.29 RY: 2.81</td>
<td>SZZ: 5.16 SY: 11.99</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARAMETER: (UNIT - POUN - INCH)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Z: 61.76 KL/R-Y: 28.34 UNL: 60.0 CMZ: 1.00 CMY: 1.00</td>
<td>CB: 0.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90 CT: 0.90 STEEL TYPE: 0.0 KS: 1.000 KV: 1.000 KBK: 1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CLAUSE</td>
</tr>
<tr>
<td>--------</td>
</tr>
<tr>
<td>TENSION</td>
</tr>
<tr>
<td>COMPRESSION</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
</tr>
<tr>
<td>SHEAR-Y</td>
</tr>
<tr>
<td>SHEAR-Z</td>
</tr>
</tbody>
</table>

---

**V. ASME NF 3000 1998 Channel**

Verify the design of channel section used for a beam member using the ASME NF 3000 1998 code.

**Details**

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.
Validation
Maximum shear force (at fixed end): $V = 51.375 \text{ kips}$ (taken from STAAD.Pro analysis)
Maximum moment (at fixed end): $M = 830.0 \text{ in-kip}$ (taken from STAAD.Pro analysis)

Slenderness Ratio
About z-axis: $(KL/r)_z = 0.85\text{(80 in)}/(5.25 \text{ in}) = 12.95$
About y-axis: $(KL/r)_y = 0.85\text{(80 in)}/(0.865 \text{ in}) = 78.61$
Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,000}{(d/ A_f)F_y} = 89.56$

As $b_f/t_f$ is not in the range of $65/\sqrt{F_y}$ and $95/\sqrt{F_y}$, the value of allowable bending compressive and tensile stress $= F_{by} = F_{cy} = 0.6 \times F_y = 21.6 \text{ ksi}$

Actual bending stress:

Bending Tensile Stress = bending compressive stress $= f_{btz} = f_{bcz} = MzD / (2I_z) = 15.4 \text{ ksi}$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear
Actual shear:

$F_y = 51.375 \text{ kips}$
$V_y = D \times t_w = 10.8 \text{ in}^2$
$f_{v,y} = F_y/V_y = 4.76 \text{ ksi}$

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in
h = clear distance flanges of the girder = $D-2t_f = 14.35 \text{ in}$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/ (a/h)^2] = 5.46$

$t = t_w = 0.72 \text{ inch}$

$C_v = 17.2$

$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4 \text{ ksi}$

Hence allowable=14.4 ksi

Therefore, Ratio $= f_v/F_v = 0.330$

Results
### Table 618: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.4</td>
<td>15.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>4.76</td>
<td>4.78</td>
<td>0.4%</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1998\ASME NF 3000 1998 Channel.std is typically installed with the program.

```
STAAD Space
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
```
STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-98) v2.0
*****************************************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
<table>
<thead>
<tr>
<th>FX</th>
<th>MY</th>
<th>MZ</th>
<th>LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ST</td>
<td>C15X50</td>
<td>(AISC SECTIONS)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PASS</td>
<td>NF-98-EQN-22</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.713</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>0.00 T</td>
<td>829999.94</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Slenderness Check:**
- Actual Ratio: 78.61
- Allowable Ratio: 300.00

**Allowable Stresses:**
- Axial: 1.55E+04
- FCZ: 2.16E+04
- FCY: 2.16E+04
- FTZ: 2.16E+04
- FTY: 2.16E+04

**Actual Stresses:**
- Axial: 0.00E+00
- FBZ: 1.54E+04
- FBY: 0.00E+00
- Shear: 4.78E+03

**Section Properties:**
- AXX: 14.70
- AYY: 10.74
- AZZ: 3.22
- RZZ: 5.24
- RYY: 0.87
- SZZ: 53.87
- SYY: 3.77

**Parameter:**
- KL/R-Z: 12.97
- KL/R-Y: 78.61
- UNL: 80.0
- CMZ: 1.00
- CMY: 1.00
- CB: 1.00
- FYLD: 36000.00
- FU: 58000.00
- Net Section Factor: 0.90
- CT: 0.75
- Steel Type: 0.0
- KS: 1.000
- KV: 1.000
- KBK: 1.000

**Critical Loads for Each Clause Check:**
<table>
<thead>
<tr>
<th>Clause</th>
<th>Ratio</th>
<th>Load</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tension</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Compression</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Comp&amp;Bend</td>
<td>0.713</td>
<td>1</td>
<td>0.00E+00</td>
<td>-</td>
<td>8.30E+05</td>
<td>0.00E</td>
</tr>
<tr>
<td>Ten&amp;Bend</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>0.00E+00</td>
<td>0.00E</td>
</tr>
<tr>
<td>Shear-Y</td>
<td>0.332</td>
<td>1</td>
<td>5.14E+04</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Shear-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>
V. ASME NF 3000 1998 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1998 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force,
\[ V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \]

Maximum moment,
\[ M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \]

Slenderness ratio = \( \frac{KL}{r} \) = 9.72 < 300

Bending About Major Axis (Z-Axis)

Section is compact: \( \frac{OD}{r} = 36.43 < 3 \times \frac{300}{F_y} = 91.67 \)

Allowable bending stress: \( F_b = 0.66 \times F_y = 23.76 \text{ ksi} \)

Actual bending stress:
\[ f_{bz} = f_{bcz} = \frac{M_z}{I_z} \left( \frac{OD}{2} \right) = \frac{331.25 \left( \frac{12.75}{2} \right)}{262} = 8.06 \text{ksi} \]

Stress ratio: 0.339

Shear

From the reference, the shear area:
\[ A_y = \frac{0.25 \left( r_2^4 - r_1^4 \right) \left( r_2 - r_1 \right)}{2 \left( r_2^3 - r_1^3 \right)} \]

where
\[ r_1 = \text{inner radius} \]
\[ r_2 = \text{outer radius} \]

Shear area: \( A_y = 6.8 \text{ in}^2 \)

Actual shear stress (at supports): \( f_v = 25.75 / 6.8 = 3.78 \text{ ksi} \)

Actual shear stress (at midspan): \( f_v = 0.75 / 6.8 = 0.110 \text{ ksi} \)

Allowable shear stress, \( F_v = 0.4 \times F_y = 14.4 \text{ ksi} \)

Stress ratio: 3.78/14.4 = 0.263

Results
Table 619: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.201</td>
<td>0.201</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1998\ASME NF 3000 1998 Pipe.std is typically installed with the program.

STAAD SPACE

START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
## Verification Examples

### V.09 Steel Design

```plaintext
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1998
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

### STAAD Output

```plaintext
STAAD.PRO CODE CHECKING - ( ASME NF3000-98) v2.0
******************************************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
FX            MY             MZ       LOCATION
=======================================================================
1 ST     PIPS120                  (AISC SECTIONS)
PASS      NF-98-EQN-22      0.339         1
0.00 T          0.00     -331249.97       25.00
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:</td>
</tr>
<tr>
<td>300.00</td>
</tr>
<tr>
<td>ALLOWABLE STRESSES: (UNIT - POUN</td>
</tr>
<tr>
<td>AXIAL: 2.12E+04 FCZ: 2.38E+04 FCY: 2.38E+04 FTZ: 2.38E+04 FTY: 2.38E+04</td>
</tr>
<tr>
<td>SHEAR: 1.44E+04</td>
</tr>
<tr>
<td>ACTUAL STRESSES: (UNIT - POUN</td>
</tr>
<tr>
<td>AXIAL: 0.00E+00 FBZ: 8.06E+03 FBY: 0.00E+00 SHEAR: 8.43E+01</td>
</tr>
<tr>
<td>----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>SECTION PROPERTIES: (UNIT -</td>
</tr>
<tr>
<td>AXX:  13.70 AYY:  7.01 AZZ:  7.01 RZZ:  4.37 RYY: 4.37</td>
</tr>
</tbody>
</table>
```

---

**Verification Examples**

**V.09 Steel Design**

```plaintext
STAAD.Pro
4194
User Manual
```
V. ASME NF 3000 1998 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1998 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) \( V = wL + P = (1 \text{ kip/in}) \times 60 \text{ in} + 2 \text{ kip} = 62 \text{ kips} \)

Maximum moment (at support) \( M = 0.5 \times w \times L^2 + P \times L = 0.5 (1 \text{ kip/in}) \times (60 \text{ in})^2 + (2 \text{ kip}) \times (60 \text{ in}) = 1,920 \text{ in-kips} \)

Slenderness Ratio

Slenderness Ratio along Z-Axis \( (KL/r)_z = 21.22 \)

Slenderness Ratio along Y-Axis \( (KL/r)_y = 50.53 \)
Maximum Slenderness Ratio = 50.53 < 300

**Bending**

**Allowable Bending Stress About Major Axis (Z Axis)**
\[
\frac{bf}{2t_f} = 4.65 > 65/\sqrt{(F_Y)} = 10.833
\]
\[
d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67
\]
Thus allowable bending stress is obtained by treating the section as compact.

\[
F_b = 0.66 \times F_Y = 23.8 \text{ ksi}
\]

**Actual Bending Stress**

Bending Tensile Stress \( f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93 \text{ ksi} < 23.8 \text{ ksi} \)

Bending Compressive Stress \( f_{bcz} = 38.7 \text{ ksi} > 23.8 \text{ ksi} \)

Critical stress ratio \( 38.7 \text{ ksi} / 23.8 \text{ ksi} = 1.626 \)

**Shear**

**Allowable Shear Stress**

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = \( D-t_f = 17.01 \text{ in} \)

Thus, \( a/h = 3.527 \) Therefore, \( k = 5.34 + \left[ \frac{4.00}{(a/h)^2} \right] = 5.65 \)

\[ t = t_w = 0.44 \text{ inch} \]

If \( Cv > 0.8 \), \( Cv = \left[ \frac{190}{(h/t)} \right] \sqrt{(h/S_y)} = 78.4 \) which is true. Thus, the value of \( Cv \) is taken as 78.4

Hence allowable shear = 14.4 ksi

**Actual Shear**

\[ V_y = D t_w = 10.68 \text{ in}^2 \]

\[ f_v = F_Y/V_y = 5.805 \text{ ksi} \]

Stress ratio = \( f_v/F_v = 0.403 \)

**Results**

**Table 620: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>50.53</td>
<td>50.53</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>(ksi)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Actual bending stress</td>
<td>38.7</td>
<td>38.5</td>
<td>0.5%</td>
<td></td>
</tr>
<tr>
<td>(ksi)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Critical shear ratio</td>
<td>1.626</td>
<td>1.620</td>
<td>0.4%</td>
<td></td>
</tr>
<tr>
<td>Parameter</td>
<td>Hand Calculation</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>------------------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>5.81</td>
<td>5.81</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.403</td>
<td>0.403</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1998\ASME NF 3000 1998 Tee.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1998
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
### Verification Examples

**V.09 Steel Design**

**STAAD Output**

```
STAAD.PRO CODE CHECKING - ( ASME NF3000-98) v2.0
*******************************************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
FX            MY             MZ       LOCATION
=======================================================================
*    1  T    W36X135                  (AISC SECTIONS)
FAIL      NF-98-EQN-22      1.620         1
0.00 T          0.00     1919999.75        0.00
|-----------------------------------------------------------------------------
| SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00          |
| ALLOWABLE STRESSES:   (UNIT - POUN INCH)             |
| AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.16E+04 FTZ: 2.16E+04 FTY: 2.16E+04 |
| SHEAR: 1.44E          |
| ACTUAL STRESSES: (UNIT - POUN INCH)             |
| AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E+03 |
|-----------------------------------------------------------------------------
| SECTION PROPERTIES: (UNIT - INCH)             |
| AXX:  19.95  AYY:   9.61  AZZ:   6.32  RZZ:  5.66  RYY: 2.37 |
| SZZ:      49.69  SYY:  18.75                        |
|-----------------------------------------------------------------------------
| PARAMETER: (UNIT - POUN INCH)             |
| KL/R-Z: 21.22  KL/R-Y: 50.53  UNL:  60.0  CMZ:  1.00  CMY: 2.37 |
| CB: 1.00  FYLD: 36000.00  FU: 58000.00  NET SECTION FACTOR: 0.90 |
| CT: 0.85  STEEL TYPE: 0.0  KS:1.000  KV:1.000  KBK: 1.000 |
```
V. ASME NF 3000 1998 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1998 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): \( V = 12.5 \text{ kips} \)

Maximum moment (at fixed end): \( M = 12.5 \text{ kips} \times 120 \text{ in} = 1,500 \text{ in kips} \)

Slenderness Ratio

\((KL/r)_z = \text{Slenderness Ratio along Z-Axis} = 18.11\)

\((KL/r)_y = \text{Slenderness Ratio along Y-Axis} = 76.68\)

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

Bending about Major Axis (Z Axis)

Allowable Bending Stress

\( f_{btz} = \frac{F_y}{0.79 - 0.002\left(b/t_f\right)\sqrt{F_y}} = 25.47 \text{ ksi} \)

Actual Bending Stress

\( f_{tz} = f_{tcz} = \frac{M/I_z}{2I_z} = 23.17 \text{ ksi} \)

Stress ratio = \( 23.16 \text{ ksi} / 25.47 \text{ ksi} = 0.909 \)

Shear
Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 120 in

h = clear distance flanges of the girder = D-2t_f = 14.91 in

Thus, a/h = 8.05, Therefore, k = 5.34 + [4.00/ (a/h)^2] = 5.36 [Ref. NF3322.6(e)(2)]

t = t_w = 0.3 inch

Cv = 1.475

Allowable shear stress = F_v = (F_y/2.89)Cv = > 0.4F_y = 14.4 ksi Hence allowable=14.4 ksi

Actual Shear Stress

F_y = 12.5 kip, V_y = D_t_w = 4.8 inch^2

f_v = F_y/V_y = 2.6 ksi

Stress ratio = f_v/F_v = 0.18

Results

Table 621: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>25.47</td>
<td>25.4</td>
<td>0.3%</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.2</td>
<td>23.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.909</td>
<td>0.910</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.6</td>
<td>2.56</td>
<td>1.5%</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.18</td>
<td>0.178</td>
<td>1.1%</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-1998\ASME NF 3000 1998 W Shaped.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-98) v2.0
******************************************************************************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
     FX   MY   MZ   LOCATION
=======================================================================
1 ST W16X40 (AISC SECTIONS) PASS NF-98-EQN-22 0.910 1
     0.00 T 0.00 -1500.00 0.00

| SLENDERNESS CHECK: ACTUAL RATIO: 76.68 ALLOWABLE RATIO: 240.00 |
| ALLOWABLE STRESSES: (UNIT - KIP INCH) |
| AXIAL: 1.49E+01 FCZ: 2.54E+01 FCY: 2.70E+01 FTZ: 2.54E+01 FTY: 2.70E+01 |
V. NF 3000-2001

V. ASME NF 3000 2001 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 2001 code.
The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36 \text{ ksi}, F_u = 58 \text{ ksi}$). Assume a net section factor of 0.9, an effective length factor of 0.85, and $C_t = 0.9$.

**Validation**

$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$

Slenderness ratio (same for both members) using the minimum value of $r$: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

**Axial Tension: Member 6**

Corrected area, $A_c = C_t \times A_x = 7.57 \text{in}^2$

Actual tensile stress $= P/A = 2,666.18 \div 7.57 = 352.2 \text{ psi}$

Allowable tensile stress (Tension capacity) $= 0.6 \times F_y = 0.6 \times 36 = 21.6 \text{ksi}$

Critical ratio $= 352.2/21,600 = 0.016$

**Axial Compression: Member 7**

Actual compressive stress $= P/A = 7,104.23 \div 8.41 = 844.74 \text{ psi}$

$C_c' = \frac{2n^2E}{Q_u Q_d F_y} = 126.2$ \hspace{1cm} (NF-3322.2(e)(5))

$(kL/r)_{\text{min}} < C_c'$, allowable compression stress, $F_a = 17.2 \text{ksi}$
Critical ratio = \( \frac{844.74}{17,200} = 0.049 \)

Results

Table 622: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member 6 (tension)</td>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>352</td>
<td>352</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>21,600</td>
<td>21,600</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Member 7 (compression)</td>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>845</td>
<td>845</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>17,200</td>
<td>17,200</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.049</td>
<td>0.049</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2001\ASME NF 3000 2001 Angle.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 2001
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>6 ST L806010 (AISC SECTIONS)</th>
<th>PASS NF-3322.1(a) 0.016 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>-2666.18 T 0.00 0.00 0.00</td>
<td></td>
</tr>
</tbody>
</table>

| SLENDERNESS CHECK: ACTUAL RATIO: 72.65 ALLOWABLE RATIO: 300.00 |
| ALLOWABLE STRESSES: (UNIT - POUN INCH) |
| AX.TENS: 2.16E+04 COMPRESS:1.72E +04 |

---

STAAD.Pro 4205 User Manual
**ACTUAL STRESSES: (UNIT - POUND INCH)**

| Ax.Tens: 3.52E+02 | Compress: 0.00E+00 |

**SECTION PROPERTIES: (UNIT - INCH)**

<table>
<thead>
<tr>
<th>Axx: 8.41</th>
<th>Ayy: 3.33</th>
<th>Azz: 2.50</th>
<th>Rzz: 1.29</th>
<th>Ryy: 2.81</th>
</tr>
</thead>
<tbody>
<tr>
<td>Szz: 5.16</td>
<td>Syy: 11.99</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**PARAMETER: (UNIT - POUND INCH)**

<table>
<thead>
<tr>
<th>Kl/R-Z: 61.76</th>
<th>Kl/R-Y: 28.34</th>
<th>Unl: 60.0</th>
<th>Cmz: 1.00</th>
<th>Cmy: 1.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cb: 0.00</td>
<td>Fyld: 36000.00</td>
<td>Fu: 58000.00</td>
<td>Net Section Factor: 0.90</td>
<td></td>
</tr>
<tr>
<td>Ct: 0.90</td>
<td>Steel Type: 0.0</td>
<td>KS: 1.000</td>
<td>KV: 1.000</td>
<td>KBK: 1.000</td>
</tr>
</tbody>
</table>

**CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUND-INCH)**

<table>
<thead>
<tr>
<th>Clause</th>
<th>Ratio</th>
<th>Load</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tension</td>
<td>0.016</td>
<td>1</td>
<td>2.67E+03</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Compression</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Comp&amp;Bend</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Ten&amp;Bend</td>
<td>0.016</td>
<td>1</td>
<td>2.67E+03</td>
<td>-</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>Shear-Y</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Shear-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td></td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>

**SLENDERNESS CHECK:**

| Actual Ratio: 61.76 | Allowable Ratio: 200.00 |

| Allowable Stresses: (UNIT - POUND INCH) |
| Ax.Tens: 2.16E+04 | Compress: 1.72E+04 |
V. ASME NF 3000 2001 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 2001 code.

Details
The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): \( V = 51.375 \text{ kips} \) (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): \( M = 830.0 \text{ in-kip} \) (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: \( (KL/r)_{z} = 0.85(80 \text{ in})/(5.25 \text{ in}) = 12.95 \)

About y-axis: \( (KL/r)_{y} = 0.85(80 \text{ in})/(0.865 \text{ in}) = 78.61 \)

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

\[
\frac{b_{f}}{t_{f}} = 4.62 < \frac{65}{\sqrt{F_{y}}} = 10.83
\]

The laterally unsupported length of the compression flange, \( L_{b} = 80 \text{ in} < \frac{76b_{f}}{\sqrt{F_{y}}} = 120 \)

and \( < \frac{20,000}{(d/A_{f})F_{y}} = 89.56 \)

As \( b_{f}/t_{f} \) is not in the range of \( 65/\sqrt{(F_{y})} \) and \( 95/\sqrt{(F_{y})} \), the value of allowable bending compressive and tensile stress = \( F_{bty} = F_{bcy} = 0.6\times F_{y} = 21.6 \text{ ksi} \)

Actual bending stress:

Bending Tensile Stress = bending compressive stress = \( f_{btz} = f_{bcz} = \frac{MzD}{(2I_{z})} = 15.4 \text{ ksi} \)

\[
830.0 \times (15.0) / (2 \times 404) = 15.41 \text{ ksi}
\]

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

\[
F_{y} = 51.375 \text{ kips}
\]

\[
V_{y} = D\times t_{w} = 10.8 \text{ in}^{2}
\]

\[
f_{v,y} = F_{y}/V_{y} = 4.76 \text{ ksi}
\]

Allowable shear:

\[ a = \text{clear distance between transverse stiffeners (here taken as length)} = 80 \text{ in} \]

\[ h = \text{clear distance flanges of the girder} = D-2t_{f} = 14.35 \text{ in} \]

Thus, \( a/h = 5.57 \), Therefore, \( k = 5.34 + [4.00/ (a/h)^{2}] = 5.46 \)

\[
t = t_{w} = 0.72 \text{ inch}
\]

\[
Cv = 17.2
\]

\[
F_{v} = (F_{y}/2.89)C_{v} \geq 0.4\times F_{y} = 14.4 \text{ ksi}
\]

Hence allowable = 14.4 ksi
Therefore, \( \text{Ratio} = \frac{f_v}{F_v} = 0.330 \)

Results

Table 623: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.4</td>
<td>15.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>4.76</td>
<td>4.78</td>
<td>0.4%</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2001\ASME NF 3000 2001 Channel.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
```
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2001
CMY 1 ALL
CMZ 1 ALL
FU 50000 ALL
FYLD 30000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE NF3000 2004
CAN 0 ALL
CMY 0.6 ALL
CMZ 0.6 ALL
CT 0.8 ALL
DFF 0 ALL
DJ1 1 ALL
DJ2 2 ALL
DMAX 39370.1 ALL
DMIN 0 ALL
FU 60000.4 ALL
FYLD 30000.2 ALL
KY 1 ALL
KZ 1 ALL
LY 0 ALL
LZ 0 ALL
MAIN 200 ALL
NSF 0.7 ALL
PROFILE W14 ALL
RATIO 1 ALL
STIFF 5 ALL
STYPE 0 ALL
TMAIN 301 ALL
TRACK 2 ALL
UNB 0 ALL
UNT 0 ALL
PRINT ANALYSIS RESULTS
FINISH
### STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>ST</td>
<td>C15X50</td>
<td>PASS</td>
<td>NF-EQN-22</td>
<td>0.713</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
<td>829999.94</td>
</tr>
</tbody>
</table>

**SLENDERNESS CHECK:** ACTUAL RATIO: 78.61 ALLOWABLE RATIO: 300.00

**ALLOWABLE STRESSES:** (UNIT - POUN INCH)
- AXIAL: $1.55E+04$ FCZ: $2.16E+04$ FCY: $2.16E+04$ FTZ: $2.16E+04$ FTY: $2.16E+04$
- SHEAR: $1.44E+04$

**ACTUAL STRESSES:** (UNIT - POUN INCH)
- AXIAL: $0.00E+00$ FBZ: $1.54E+04$ FBY: $0.00E+00$ SHEAR: $4.78E+03$

**SECTION PROPERTIES:** (UNIT - INCH)
- $A_{XX}: 14.70$ $A_{YY}: 10.74$ $A_{ZZ}: 3.22$ $R_{ZZ}: 5.24$ $R_{YY}: 0.87$
- $S_{ZZ}: 53.87$ $S_{YY}: 3.77$

**PARAMETER:** (UNIT - POUN INCH)
- $K_{L/R-Z}: 12.97$ $K_{L/R-Y}: 78.61$ $U_{NL}: 80.0$ $C_{MZ}: 1.00$ $C_{MY}: 1.00$
- $C_{B}: 1.00$ $F_YLD: 36000.00$ $F_U: 58000.00$ $NET\ SECTION\ FACTOR: 0.90$
- $C_{T}: 0.75$ STEEL TYPE: $0.0$ $K_{S}: 1.000$ $K_{V}: 1.000$ $K_{B}: 1.000$

**CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH)**
- **CLAUSE** | **RATIO** | **LOAD** | **FX** | **VY** | **VZ** | **MZ** |
- **TENSION** | $0.000$ | $0$ | $0.00E+00$ | - | - | - |
- **COMPRESSION** | $0.000$ | $0$ | $0.00E+00$ | - | - | - |
- **COMP&BEND** | $0.713$ | $1$ | $0.00E+00$ | - | - | $8.30E+05$ $0.00E+00$
Verify the design of a pipe section used for a beam member per the ASME NF 3000 2001 code.

Details
The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force,
\[ V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \]

Maximum moment,
\[ M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \]

Slenderness ratio = \( KL/r \) = 9.72 < 300

Bending About Major Axis (Z-Axis)

Section is compact: \( \frac{OD}{r} = 36.43 < 3 \frac{300}{F_y} = 91.67 \)

Allowable bending stress: \( F_b = 0.66 \times F_y = 23.76 \text{ksi} \)

Actual bending stress:
\[ f_{btz} = f_{bcz} = \frac{M_z}{I_z} \left( \frac{OD}{2} \right)^2 = \frac{331.25}{262} \left( \frac{12.75}{2} \right) = 8.06 \text{ksi} \]

Stress ratio: 0.339

Shear

From the reference, the shear area:
\[ A_y = \frac{0.25n(r_2^4 - r_1^4)}{2(r_2^3 - r_1^3)} \]

where
\[ r_1 = \text{inner radius} \]
\[ r_2 = \text{outer radius} \]

Shear area: \( A_y = 6.8 \text{ in}^2 \)

Actual shear stress (at supports): \( f_v = 25.75 / 6.8 = 3.78 \text{ ksi} \)

Actual shear stress (at midspan): \( f_v = 0.75 / 6.8 = 0.110 \text{ ksi} \)
Allowable shear stress, \( F_v = 0.4 \times F_y = 14.4 \text{ ksi} \)

Stress ratio: \( \frac{3.78}{14.4} = 0.263 \)

Results

Table 624: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress at mid-span (ksi)</td>
<td>0.084</td>
<td>0.0843</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio at supports</td>
<td>0.201</td>
<td>0.201</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2001\ASME NF 3000 2001 Pipe.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN  
1 TABLE ST PIPS120  
CONSTANTS  
MATERIAL STEEL ALL  
SUPPORTS  
1 PINNED  
2 FIXED BUT FX MY MZ  
LOAD 1 LOADTYPE None TITLE LOAD CASE 1  
MEMBER LOAD  
1 CON GY -1500 25 0  
1 UNI GY -1000 0 50  
PERFORM ANALYSIS  
PARAMETER 1  
CODE NF3000 2001  
CMY 1 ALL  
CMZ 1 ALL  
CT 0.85 ALL  
FU 58000 ALL  
FYLD 36000 ALL  
KY 0.85 ALL  
KZ 0.85 ALL  
UNT 50 ALL  
UNB 50 ALL  
MAIN 200 ALL  
NSF 0.9 ALL  
RATIO 1 ALL  
TMAIN 300 ALL  
UNB 25 ALL  
UNT 25 ALL  
TRACK 2 ALL  
CHECK CODE ALL  
FINISH

STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-01) v2.0  
********************************************  
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)  
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/  
FX MY MZ LOCATION  
=======================================================================  
1  ST PIPS120 (AISC SECTIONS) PASS NF-EQN-22 0.339 1  
0.00 T 0.00 -331249.97 25.00  
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SLENDERNESS CHECK: ACTUAL RATIO: 9.72 ALLOWABLE RATIO: 300.00</td>
</tr>
<tr>
<td>ALLOWABLE STRESSES: (UNIT - POUN INCH)</td>
</tr>
<tr>
<td>AXIAL: 2.12E+04 FCZ: 2.38E+04 FCY: 2.38E+04 FTZ: 2.38E+04 FTY: 2.38E+04</td>
</tr>
<tr>
<td>SHEAR: 1.44E+04</td>
</tr>
<tr>
<td>ACTUAL STRESSES: (UNIT - POUN INCH)</td>
</tr>
</tbody>
</table>

Verification Examples
V.09 Steel Design

STAAD.Pro 4214 User Manual
V. ASME NF 3000 2001 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 2001 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation
Maximum shear force (at support) = \( V = wL + P = (1 \text{ kip/in}) \times 60 \text{ in} + 2 \text{ kip} = 62 \text{ kips} \)

Maximum moment (at support) = \( M = 0.5 \times w \times L^2 + P \times L = 0.5 (1 \text{ kip/in}) \times (60 \text{ in})^2 + (2 \text{ kip}) \times (60 \text{ in}) = 1,920 \text{ in kips} \)

Slenderness Ratio

Slenderness Ratio along Z-Axis = \( (KL/r)_z = 21.22 \)

Slenderness Ratio along Y-Axis = \( (KL/r)_y = 50.53 \)

Maximum Slenderness Ratio = 50.53 < 300

Bending

Allowable Bending Stress About Major Axis (Z Axis)

\[
\frac{b f}{2 t f} = 4.65 > \frac{65}{\sqrt{F_Y}} = 10.833
\]

\[
d/t_w = 28.3 < \frac{640}{\sqrt{F_Y}} = 106.67
\]

Thus allowable bending stress is obtained by treating the section as compact.

\[
F_b = 0.66 \times F_Y = 23.8 \text{ ksi}
\]

Actual Bending Stress

Bending Tensile Stress = \( f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93 \text{ ksi} < 23.8 \text{ ksi} \)

Bending Compressive Stress = \( f_{bcz} = 38.7 \text{ ksi} > 23.8 \text{ ksi} \)

Critical stress ratio = \( 38.7 \text{ ksi} / 23.8 \text{ ksi} = 1.626 \)

Shear

Allowable Shear Stress

\[
a = \text{clear distance between transverse stiffeners (here taken as length)} = 60 \text{ in}
\]

\[
h = \text{clear distance flanges of the girder} = D-t_f = 17.01 \text{ in}
\]

Thus, \( a/h = 3.527 \) Therefore, \( k = 5.34 + [4.00/ (a/h)^2] = 5.65 \)

\[
t = t_w = 0.44 \text{ inch}
\]

If \( Cv > 0.8, \ Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4 \) which is true. Thus, the value of \( Cv \) is taken as 78.4

Allowable shear stress = \( F_V = (F_Y/2.89)Cv \leq 0.4F_Y = 14.4 \text{ ksi} \)

Hence allowable shear = 14.4ksi

Actual Shear

\[
V_y = Dt_w = 10.68 \text{ in}^2
\]

\[
f_v = F_Y/V_y = 5.805 \text{ ksi}
\]

Stress ratio = \( f_v/F_V = 0.403 \)

Results

Table 625: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>50.53</td>
<td>50.53</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
## Parameter

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>38.7</td>
<td>38.5</td>
<td>0.5%</td>
<td></td>
</tr>
<tr>
<td>Critical shear ratio</td>
<td>1.626</td>
<td>1.620</td>
<td>0.4%</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>5.81</td>
<td>5.81</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.403</td>
<td>0.403</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2001\ASME NF 3000 2001 Tee.std is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
```
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2001
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
</tr>
<tr>
<td>* 1 T</td>
<td>W36X135</td>
<td>FAIL</td>
<td>NF-EQN-22</td>
<td>1.620</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
<td>1919999.75</td>
<td>0.00</td>
</tr>
</tbody>
</table>

| SLENDERNESS CHECK: | ACTUAL RATIO: 50.53 ALLOWABLE RATIO: 300.00 |
| ALLOWABLE STRESSES: | (UNIT - POUN INCH) |
| AXIAL: 1.16E+04 | FCZ: 2.38E+04 | FCY: 2.16E+04 | FTZ: 2.16E+04 | FTY: 2.16E+04 |
| SHEAR: 1.44E+04 |
| ACTUAL STRESSES: | (UNIT - POUN INCH) |
| AXIAL: 0.00E+00 | FBZ: 3.85E+04 | FBY: 0.00E+00 | SHEAR: 5.81E+03 |

| SECTION PROPERTIES: | (UNIT - INCH) |
| AXX: 19.95 | AYY: 9.61 | AZZ: 6.32 | RZZ: 5.66 | RYY: 2.37 |
| SZZ: 49.69 | SYY: 18.75 | |

staad.pro code checking - (ASME NF3000-01) v2.0
*******************************************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
*******************************************************************************

STAAD.Pro 4218 User Manual
V. ASME NF 3000 2001 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 2001 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5 \text{kips}$

Maximum moment (at fixed end): $M = 12.5 \text{kips} \times 120 \text{ in} = 1,500 \text{ in kips}$

Slenderness Ratio

$$(K_{L/r})_z = \text{Slenderness Ratio along Z-Axis} = 18.11$$

$$(K_{L/r})_y = \text{Slenderness Ratio along Y-Axis} = 76.68$$

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

Bending about Major Axis (Z Axis)

Allowable Bending Stress

$$\frac{b_f}{2t} = 6.862 < 65/(F_y)^{0.5} \text{ [Ref. NF3322.1(d)(2)]}$$
Thus, allowable bending tensile stress \( F_{btz} = F_y \times \{0.79 - 0.002 \times (b_f/2t_f) \times \sqrt{F_y} \} = 25.47 \text{ksi} \)

Actual Bending Stress

Bending Tensile Stress = Bending Compressive Stress \( f_{btz} = f_{bcz} = M_z/D/2I_z = 23.17 \text{ksi} \)

Stress ratio = \( 23.16 \text{ksi} / 25.47 \text{ksi} = 0.909 \)

Shear

Allowable Shear Stress

\( a = \) clear distance between transverse stiffeners (here taken as length) = 120 in

\( h = \) clear distance flanges of the girder = \( D-2t_f = 14.91 \text{in} \)

Thus, \( a/h = 8.05 \), Therefore, \( k = 5.34 + \{4.00/ (a/h)^2\} = 5.36 \) [Ref. NF3322.6(e)(2)]

\[ t = t_w = 0.3 \text{inch} \]

\[ Cv = 1.475 \]

Allowable shear stress \( F_v = (F_y/2.89)Cv = > 0.4F_y = 14.4 \text{ksi} \) Hence allowable = 14.4 ksi

Actual Shear Stress

\( F_y = 12.5 \text{kip}, \ V_y = D t_w = 4.8 \text{inch}^2 \)

\( f_v = F_y/V_y = 2.6 \text{ksi} \)

Stress ratio = \( f_v/F_v = 0.18 \)

Results

Table 626: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>25.47</td>
<td>25.4</td>
<td>0.3%</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.2</td>
<td>23.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.909</td>
<td>0.910</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.6</td>
<td>2.56</td>
<td>1.5%</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.18</td>
<td>0.178</td>
<td>1.1%</td>
<td></td>
</tr>
</tbody>
</table>
The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2001\ASME NF 3000 2001 W Shaped.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
END MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 2001
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/LOCATION</th>
<th>CRITICAL COND/LOCATION</th>
<th>RATIO/LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ST W16X40</td>
<td>PASS NF-EQN-22</td>
<td>0.910</td>
<td>1</td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.09 Steel Design

**Slenderness Check**
- **Actual Ratio:** 76.68
- **Allowable Ratio:** 240.00

**Allowable Stresses** (Unit - KIP Inch)
- **Axial:** 1.49E+01
- **Shear:** 1.44E+01

**Actual Stresses** (Unit - KIP Inch)
- **Axial:** 0.00E+00
- **Shear:** 2.56E+00

**Section Properties** (Unit - Inch)
- **Axx:** 11.80
- **Ayy:** 4.88
- **Az:** 4.71
- **Rzz:** 6.63
- **Ryy:** 1.56
- **Szz:** 64.75
- **Syy:** 8.26

**Parameter** (Unit - KIP Inch)
- **Kl/R-Z:** 18.11
- **Kl/R-Y:** 76.68
- **Unl:** 72.0
- **Cmz:** 0.85
- **Cmy:** 0.85
- **Cb:** 1.00
- **Fyld:** 36.00
- **Fu:** 58.02
- **Net Section Factor:** 1.00
- **St:** 0.75
- **Steel Type:** 0.0
- **Ks:** 1.000
- **Kv:** 1.000
- **Kbk:** 1.000

**Critical Loads for Each Clause Check** (Units KIP - Inch)

<table>
<thead>
<tr>
<th>Clause</th>
<th>Ratio</th>
<th>Load</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tension</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Compression</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Comp&amp;Bend</td>
<td>0.910</td>
<td>1</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>1.50E+03 0.00E</td>
</tr>
<tr>
<td>Ten&amp;Bend</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>0.00E+00 0.00E</td>
</tr>
<tr>
<td>Shear-Y</td>
<td>0.178</td>
<td>1</td>
<td>-</td>
<td>1.25E+01</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Shear-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro 4222 User Manual**
**V. NF 3000-2004**

**V. ASME NF 3000 2004 Angle**

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 2004 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

![Figure 453: Space frame subjected to static loads](image)

Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36$ ksi, $F_u = 58$ ksi).

Assume a net section factor of 0.9, an effective length factor of 0.85, and $C_t = 0.9$.

Validation

\[
L = \sqrt{5^2 + 6^2} \text{(12in/ft)} = 93.726\text{in}
\]

Slenderness ratio (same for both members) using the minimum value of \(r\): $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, \(kL/r_z = 61.76 < 300\)

For member 7, \(kL/r_z = 61.76 < 200\)

Axial Tension: Member 6

Corrected area, $A_c = C_t \times A_x = 7.57 \text{ in}^2$
Actual tensile stress = \( P/A = 2,666.18 / 7.57 = 352.2 \text{ psi} \)

Allowable tensile stress (Tension capacity) = \( 0.6 \times F_y = 0.6 \times 36 = 21.6 \text{ ksi} \)

Critical ratio = \( 352.2 / 21,600 = 0.016 \)

**Axial Compression: Member 7**

Actual compressive stress = \( P/A = 7,104.23 / 8.41 = 844.74 \text{ psi} \)

\[
C'_c = \sqrt{\frac{2\pi^2E}{\alpha_s \alpha_a F_y}} = 126.2
\]  

\((kL/r)_{\text{min}} < C'_c\), allowable compression stress, \( F_a = 17.2 \text{ ksi} \)

Critical ratio = \( 844.74 / 17,200 = 0.049 \)

Results

**Table 627: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member 6 (tension)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>352</td>
<td>352</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>21,600</td>
<td>21,600</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.016</td>
<td>0.016</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Member 7 (compression)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slenderness ratio</td>
<td>61.76</td>
<td>61.76</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual tensile stress (psi)</td>
<td>845</td>
<td>845</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable tensile stress (psi)</td>
<td>17,200</td>
<td>17,200</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.049</td>
<td>0.049</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2004\ASME NF 3000 2004 Angle.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND

JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;

MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;

DEFINE MATERIAL START

ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03

TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2

END DEFINE MATERIAL

MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010

CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED

MEMBER TRUSS
6 7

LOAD 1 LOADTYPE None TITLE LOAD CASE 1

JOINT LOAD
4 FX 7500
6 FY -10000

PERFORM ANALYSIS

PRINT MEMBER PROPERTIES LIST 6 7

SECTION 0 0.25 0.5 0.75 1 MEMB 6 7

PRINT MEMBER SECTION FORCES LIST 6 7

PRINT MEMBER FORCES LIST 6 7

PRINT MEMBER STRESSES LIST 6

PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
MAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>6</th>
<th>ST</th>
<th>L806010</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>PASS</td>
<td>NF-3322.1(a)</td>
<td>0.016</td>
</tr>
</tbody>
</table>
SLENDERNESS CHECK: ACTUAL RATIO: 72.65 ALLOWABLE RATIO: 300.00
ALLOWABLE STRESSES: (UNIT - POUND INCH)
| AX. TENS: 2.16E+04 | COMPRESS: 1.72E 

ACTUAL STRESSES: (UNIT - POUND INCH)
| AX. TENS: 3.52E+02 | COMPRESS: 0.00E 

SECTION PROPERTIES: (UNIT - INCH)
| AXX: 8.41 AYY: 3.33 AZZ: 2.50 RZZ: 1.29 RYY: 2.81 SZZ: 5.16 SYY: 11.99 |

PARAMETER: (UNIT - POUND INCH)
| KL/R-Z: 61.76 KL/R-Y: 28.34 UNL: 60.0 CMZ: 1.00 CMY: 1.00 |
| CB: 0.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90 |
| CT: 0.90 STEEL TYPE: 0.0 KS:1.000 KV:1.000 KBK: 1.000 |

CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUND-INCH)
<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.016</td>
<td>1</td>
<td>2.67E+03</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>0.00E+00 0.00E</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
<td>0.016</td>
<td>1</td>
<td>2.67E+03</td>
<td>-</td>
<td>-</td>
<td>0.00E+00 0.00E</td>
</tr>
<tr>
<td>SHEAR-Y</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>
### Verification Examples

#### V.09 Steel Design

<table>
<thead>
<tr>
<th>7 ST L806010 (AISC SECTIONS)</th>
<th>PASS</th>
<th>NF-3322.1(c)</th>
<th>0.049</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>7104.23 C</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Slenderness Check:</th>
<th>Actual Ratio: 61.76 Allowable Ratio: 200.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable Stresses:</td>
<td>(Unit - Pound Inch)</td>
</tr>
<tr>
<td>Ax.Tens: 2.16E+04</td>
<td>Compress: 1.72E</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Actual Stresses:</th>
<th>(Unit - Pound Inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ax.Tens: 0.00E+00</td>
<td>Compress: 8.45E</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Section Properties:</th>
<th>(Unit - Inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>AXX: 8.41 AYY: 3.33 AZZ: 2.50 RZZ: 1.29 RYY: 2.81 SZZ: 5.16 SYY: 11.99</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter:</th>
<th>(Unit - Pound Inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>KL/R-Z: 61.76 KL/R-Y: 28.34 UNL: 60.0 CMZ: 1.00 CMY: 1.00</td>
<td></td>
</tr>
<tr>
<td>CB: 0.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90</td>
<td></td>
</tr>
<tr>
<td>CT: 0.90 STEEL TYPE: 0.0 KS: 1.000 KV: 1.000 KBK: 1.000</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Critical Loads for Each Clause Check (Units Pound-Inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause</td>
</tr>
<tr>
<td>--------</td>
</tr>
<tr>
<td>Tension</td>
</tr>
<tr>
<td>Compression</td>
</tr>
<tr>
<td>Comp&amp;Bend</td>
</tr>
<tr>
<td>Ten&amp;Bend</td>
</tr>
<tr>
<td>Shear-Y</td>
</tr>
<tr>
<td>Shear-Z</td>
</tr>
</tbody>
</table>
**V. ASME NF 3000 2004 Channel**

Verify the design of channel section used for a beam member using the ASME NF 3000 2004 code.

**Details**

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

**Validation**

Maximum shear force (at fixed end): \( V = 51.375 \text{ kips} \) (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): \( M = 830.0 \text{ in-kip} \) (taken from STAAD.Pro analysis)

**Slenderness Ratio**

About z-axis: \( (KL/r)_z = 0.85(80 \text{ in})/(5.25 \text{ in}) = 12.95 \)

About y-axis: \( (KL/r)_y = 0.85(80 \text{ in})/(0.865 \text{ in}) = 78.61 \)

Max. slenderness ratio < 300, OK.

**Bending About Major Axis (Z Axis)**

\[
\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83
\]

The laterally unsupported length of the compression flange, \( L_b = 80 < \frac{76b_f}{\sqrt{F_y}} = 120 \)

and \( < \frac{20,000}{(d/A_f)F_y} = 89.56 \)

As \( b_f/t_f \) is not in the range of \( 65/\sqrt{(F_y)} \) and \( 95/\sqrt{(F_y)} \), the value of allowable bending compressive and tensile stress = \( F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6 \text{ ksi} \)

Actual bending stress:

Bending Tensile Stress = bending compressive stress = \( f_{btz} = f_{bcz} = \frac{MzD}{(2I_z)} = 15.4 \text{ ksi} \)

\[
830.0 \text{ (15.0) } / (2 \times 404) = 15.41 \text{ ksi}
\]

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

**Shear**

Actual shear:

\[
F_y = 51.375 \text{ kips} \\
V_y = D \times t_w = 10.8 \text{ in}^2 \\
f_{v,y} = F_y/V_y = 4.76 \text{ ksi}
\]

Allowable shear:
a = clear distance between transverse stiffeners (here taken as length) = 80 in
h = clear distance flanges of the girder = D-2t = 14.35 in
Thus, a/h = 5.57, Therefore, k = 5.34 + [4.00/ (a/h)²] = 5.46

\[ t = t_w = 0.72 \text{ inch} \]

\[ C_v = 17.2 \]

\[ F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4 \text{ ksi} \]

Hence allowable=14.4 ksi
Therefore, Ratio = \( f_v/F_v = 0.330 \)

Results

Table 628: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>78.61</td>
<td>78.61</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>21.6</td>
<td>21.6</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>15.4</td>
<td>15.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical ratio</td>
<td>0.713</td>
<td>0.713</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>4.76</td>
<td>4.78</td>
<td>0.4%</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2004\ASME NF 3000 2004 Channel.std is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/LOCATION</th>
<th>CRITICAL COND/LOADING/</th>
<th>RATIO/</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
</tr>
<tr>
<td>0.00</td>
<td>0.00</td>
<td>829999.94</td>
<td>1</td>
<td>0.713</td>
<td>1</td>
</tr>
</tbody>
</table>

SLENDERNESS CHECK: ACTUAL RATIO: 78.61 ALLOWABLE RATIO: 300.00
ALLOWABLE STRESSES: (UNIT - POUN INCH)
AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
V. ASME NF 3000 2004 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 2004 code.

Details
The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, \( V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 \text{kips} \)

Maximum moment, \( M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 \text{in} \cdot \text{kip} \)

Slenderness ratio = \( (KL/r) = 9.72 < 300 \)

Bending About Major Axis (Z-Axis)

Section is compact: \( \frac{OD}{t} = 36.43 \times \frac{300}{F_y} = 91.67 \)

Allowable bending stress: \( F_b = 0.66 \times F_y = 23.76 \text{ksi} \)

Actual bending stress: \( f_{btz} = f_{bcz} = \frac{Mz}{Iz} \left( \frac{OD}{2} \right) = \frac{331.25 \left( \frac{12.75}{2} \right)}{262} = 8.06 \text{ksi} \)

Stress ratio: 0.339

Shear

From the reference, the shear area:

\[
A_y = \frac{0.25\pi r_2^4 - r_1^4}{2\pi (r_2^3 - r_1^3)}
\]

where

\( r_1 = \text{inner radius} \)
\( r_2 = \text{outer radius} \)

Shear area: \( A_y = 6.8 \text{ in}^2 \)

Actual shear stress (at supports): \( f_v = 25.75 / 6.8 = 3.78 \text{ ksi} \)

Actual shear stress (at midspan): \( f_v = 0.75 / 6.8 = 0.110 \text{ ksi} \)

Allowable shear stress, \( F_v = 0.4 \times F_y = 14.4 \text{ ksi} \)

Stress ratio: 3.78/14.4 = 0.263

Results

Table 629: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>9.72</td>
<td>9.72</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>8.06</td>
<td>8.06</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Parameter</td>
<td>Hand Calculation</td>
<td>STAAD.Pro</td>
<td>Difference</td>
<td>Comments</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>------------------</td>
<td>-----------</td>
<td>------------</td>
<td>----------</td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.339</td>
<td>0.339</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress at mid-span (ksi)</td>
<td>0.084</td>
<td>0.0843</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio at supports</td>
<td>0.201</td>
<td>0.201</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2004\ASME NF 3000 2004 Pipe.std is typically installed with the program.

STAAD SPACE

START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/CRITICAL COND/LOADING/</th>
<th>RATIO/LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ST</td>
<td>PIPS120</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>PASS</td>
<td>NF-EQN-22</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.339</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td>-331249.97</td>
<td>25.00</td>
</tr>
</tbody>
</table>

**SLENDERNESS CHECK:**
- **ACTUAL RATIO:** 9.72
- **ALLOWABLE RATIO:** 300.00

**ALLOWABLE STRESSES:**
- **AXIAL:** 2.12E+04
- **FCZ:** 2.38E+04
- **FCY:** 2.38E+04
- **FTZ:** 2.38E+04
- **FTY:** 2.38E+04
- **SHEAR:** 1.44E+04

**ACTUAL STRESSES:**
- **AXIAL:** 0.00E+00
- **FBZ:** 8.06E+03
- **FBEY:** 0.00E+00
- **SHEAR:** 8.43E+01

**SECTION PROPERTIES:**
- **AXX:** 13.70
- **AYY:** 7.01
- **AZZ:** 7.01
- **RZZ:** 4.37
- **RYY:** 4.37
- **SZZ:** 41.10
- **SYY:** 41.10

**PARAMETER:**
- **UNIT - POUN INCH**
V. ASME NF 3000 2004 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 2004 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = \( V = wL + P = (1 \text{ kip/in}) \times 60 \text{ in} + 2 \text{ kip} = 62 \text{ kips} \)

Maximum moment (at support) = \( M = 0.5 \times w \times L^2 + P \times L = 0.5 (1 \text{ kip/in}) \times (60 \text{ in})^2 + (2 \text{ kip}) \times (60 \text{ in}) = 1,920 \text{ in\ kips} \)

Slenderness Ratio

Slenderness Ratio along Z-Axis = \((KL/r)_z = 21.22\)

Slenderness Ratio along Y-Axis = \((KL/r)_y = 50.53\)

Maximum Slenderness Ratio = 50.53 < 300

Bending

Allowable Bending Stress About Major Axis (Z Axis)

\[ \frac{b'_f}{2t_f} = 4.65 > 65/\sqrt{(F_Y)} = 10.833 \]
\[ \frac{d}{t_w} = 28.3 < \frac{640}{\sqrt{F_Y}} = 106.67 \]

Thus allowable bending stress is obtained by treating the section as compact.

\[ F_b = 0.66 \times F_Y = 23.8 \text{ ksi} \]

Actual Bending Stress

Bending Tensile Stress \( f_{btz} = \left(\frac{Mz}{Iz}\right) \times \bar{Y} = 14.93 \text{ ksi} < 23.8 \text{ ksi} \)

Bending Compressive Stress \( f_{bcz} = 38.7 \text{ ksi} > 23.8 \text{ ksi} \)

Critical stress ratio \( = \frac{38.7 \text{ ksi}}{23.8 \text{ ksi}} = 1.626 \)

Shear

Allowable Shear Stress

\[ a = \text{clear distance between transverse stiffeners (here taken as length)} = 60 \text{ in} \]

\[ h = \text{clear distance flanges of the girder} = D-t_f = 17.01 \text{ in} \]

Thus, \( \frac{a}{h} = 3.527 \) Therefore, \( k = 5.34 + \frac{4.00}{(a/h)^2} = 5.65 \)

\[ t = t_w = 0.44 \text{ inch} \]

If \( Cv > 0.8, \ C_v = \left(\frac{190}{(h/t)}\right) \sqrt{(k/S_y)} = 78.4 \) which is true. Thus, the value of \( C_v \) is taken as 78.4

Allowable shear stress \( = F_v = \left(\frac{F_Y}{2.89}\right) C_v \leq 0.4 F_Y = 14.4 \text{ ksi} \)

Hence allowable shear = 14.4 ksi

Actual Shear

\[ V_y = D t_w = 10.68 \text{ in}^2 \]

\[ f_v = \frac{F_y}{V_y} = 5.805 \text{ ksi} \]

Stress ratio \( = \frac{f_v}{F_v} = 0.403 \)

Results

**Table 630: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>50.53</td>
<td>50.53</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>38.7</td>
<td>38.5</td>
<td>0.5%</td>
<td></td>
</tr>
<tr>
<td>Critical shear ratio</td>
<td>1.626</td>
<td>1.620</td>
<td>0.4%</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>5.81</td>
<td>5.81</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
### Parameter

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear stress ratio</td>
<td>0.403</td>
<td>0.403</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD Input

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2004\ASME NF 3000 2004 Tee.std` is typically installed with the program.

```
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOIN COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
```
STAAD Output

STAAD.PRO CODE CHECKING - ( ASME NF3000-04) v2.0
***********************************************************************
ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/ LOCATION
FX MY MZ
=======================================================================
* 1 T W36X135                  (AISC SECTIONS)
FAIL NF-EQN-22         1.620         1
0.00 T 0.00 1919999.75 0.00

<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SLENDERNESS CHECK: ACTUAL RATIO: 50.53 ALLOWABLE RATIO: 300.00</td>
</tr>
<tr>
<td>ALLOWABLE STRESSES: (UNIT - POUN INCH)</td>
</tr>
<tr>
<td>AXIAL: 1.16E+04 FCZ: 2.38E+04 FCY: 2.16E+04 FTZ: 2.16E+04 FTY: 2.16E+04</td>
</tr>
<tr>
<td>SHEAR: 1.44E+04</td>
</tr>
<tr>
<td>ACTUAL STRESSES: (UNIT - POUN INCH)</td>
</tr>
<tr>
<td>AXIAL: 0.00E+00 FBZ: 3.85E+04 FBY: 0.00E+00 SHEAR: 5.81E+03</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>SECTION PROPERTIES: (UNIT - INCH)</td>
</tr>
<tr>
<td>AXX: 19.95 AYY: 9.61 AZZ: 6.32 RZZ: 5.66 RYY: 2.37</td>
</tr>
<tr>
<td>SZZ: 49.69 SYY: 18.75</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>PARAMETER: (UNIT - POUN INCH)</td>
</tr>
<tr>
<td>KL/R-Z: 21.22 KL/R-Y: 50.53 UNL: 60.0 CMZ: 1.00 CMY: 1.00</td>
</tr>
<tr>
<td>CB: 1.00 FYLD: 36000.00 FU: 58000.00 NET SECTION FACTOR: 0.90</td>
</tr>
<tr>
<td>CT: 0.85 STEEL TYPE: 0.0 KS:1.000 KV:1.000 KBK: 1.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-INCH)</td>
</tr>
<tr>
<td>CLAUSE RATIO LOAD FX VY VZ MZ</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
### V. ASME NF 3000 2004 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 2004 code.

**Details**

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

**Validation**

**Maximum shear force (constant shear):** \( V = 12.5 \text{ kips} \)

**Maximum moment (at fixed end):** \( M = 12.5 \text{ kips} \times 120 \text{ in} = 1,500 \text{ in·kips} \)

**Slenderness Ratio**

\[ (KL/r)_z = \text{Slenderness Ratio along Z-Axis} = 18.11 \]

\[ (KL/r)_y = \text{Slenderness Ratio along Y-Axis} = 76.68 \]

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

**Bending about Major Axis (Z Axis)**

**Allowable Bending Stress**

\[ b_t/2t_f = 6.862 < 65/(F_y)^{0.5} \text{ [Ref. NF3322.1(d)(2)]} \]

Thus, allowable bending tensile stress = \( F_{btz} = F_y \times [0.79 - 0.002 \times (b_t/2t_f) \times \sqrt{F_y}] = 25.47 \text{ ksi} \)

**Actual Bending Stress**

Bending Tensile Stress = Bending Compressive Stress = \( f_{btz} = f_{bcz} = M_zD/2I_z = 23.17 \text{ ksi} \)

**Stress ratio = 23.16 ksi / 25.47 ksi = 0.909**

**Shear**

**Allowable Shear Stress**

\[ a = \text{clear distance between transverse stiffeners (here taken as length)} = 120 \text{ in} \]

\[ h = \text{clear distance flanges of the girder} = D - 2t_f = 14.91 \text{ in} \]

Thus, \( a/h = 8.05, \text{ Therefore, } k = 5.34 + [4.00/ (a/h)^2] = 5.36 \text{ [Ref. NF3322.6(e)(2)]} \)
\( t = t_w = 0.3 \text{ inch} \)

\[ C_v = 1.475 \]

Allowable shear stress \( F_v = (F_y/2.89)C_v = > 0.4F_y = 14.4 \text{ ksi} \) Hence allowable=14.4ksi

Actual Shear Stress

\[ F_y = 12.5 \text{ kip}, \ V_y = Dt_w = 4.8 \text{ inch}^2 \]

\[ f_v = F_y/V_y = 2.6 \text{ ksi} \]

Stress ratio \( = f_v/F_v = 0.18 \)

Results

**Table 631: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>76.68</td>
<td>76.68</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress (ksi)</td>
<td>25.47</td>
<td>25.4</td>
<td>0.3%</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress (ksi)</td>
<td>23.2</td>
<td>23.2</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Critical stress ratio</td>
<td>0.909</td>
<td>0.910</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Allowable shear stress (ksi)</td>
<td>14.4</td>
<td>14.4</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual shear stress (ksi)</td>
<td>2.6</td>
<td>2.56</td>
<td>1.5%</td>
<td></td>
</tr>
<tr>
<td>Shear stress ratio</td>
<td>0.18</td>
<td>0.178</td>
<td>1.1%</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2004\ASME NF 3000 2004 W Shaped.std is typically installed with the program.

**STAAD SPACE**

START JOB INFORMATION

ENGINEER DATE 18-Jun-18

END JOB INFORMATION

SET SHEAR

UNIT FEET KIP

JOINT COORDINATES

1 0 0 0; 2 10 0 0;

MEMBER INCIDENCES

1 1 2;

MEMBER PROPERTY AMERICAN

1 TABLE ST W16X40

UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 2004
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH

STAAD Output

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>ST W16X40</td>
<td>PASS</td>
<td>NF-EQN-22</td>
<td>0.910</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
<td>-1500.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
</tr>
</tbody>
</table>

SLENDERNESS CHECK: ACTUAL RATIO: 76.68 ALLOWABLE RATIO: 240.00
ALLOWABLE STRESSES: (UNIT - KIP INCH)
| AXIAL: 1.49E+01 | FCZ: 2.54E+01 | FCY: 2.70E+01 | FTZ: 2.54E+01 | FTY: 2.70E+01 |
| SHEAR: 1.44E+01 |

ACTUAL STRESSES: (UNIT - KIP INCH)
| AXIAL: 0.00E+00 | FBZ: 2.32E+01 | FBY: 0.00E+00 | SHEAR: 2.56E+00 |

Verification Examples
V.09 Steel Design

STAAD.Pro
4241
User Manual
V.ASME NF 3000 2004 STYPE 1 Pipe

Verify the design results an austenitic stainless steel pipe section, designed as per ASME NF 3000 2004.

Details

A beam (beam no. 7 in the model) of length 10 ft is loaded with a uniformly distributed load of 1 kip/ft over the whole span, and axial compressive loads of 100 kip at both ends. The beam is assigned with a steel pipe section of PIPS120 and is designed in accordance with ASME NF3000 2004.

The austenitic stainless steel has a yield strength of 36 ksi, an ultimate rupture strength of 58 ksi, and a modulus of elasticity of 29,000 ksi.

Validation
From the analysis:

\[ F_x = 100.145 \text{ kip} \]
\[ F_y = 5 \text{ kip} \text{ (at ends)} \]
\[ M_z = 97.683 \text{ in} \cdot \text{k} \text{ (at mid-span)} \]
\[ M_y = 10.798 \text{ in} \cdot \text{k} \]

\((KL/r)_z = \text{Slenderness Ratio along Z-Axis} = 27.44 = (KL/r)_y = \text{Slenderness Ratio along Y-Axis < 200 Hence OK (Refer NF-3322.2(c)(1))}\)

**Check Against Axial Compression**

As per clause NF 3322.2(d)(2)(c),

\[ \frac{OD}{t_w} = 12.75/0.35 = 36.429 < 3,300/F_y = 91.667 \]

Since \((KL/r)_y = 27.44 = (kl/r)_z < 120\)

As per NF 3322.1(c)(2)(a),

Allowable compressive stress, \(F_a = F_y \left(0.47 - \frac{kl}{r}444\right) = 14.7 \text{ ksi}\)

Actual compressive stress \(f_a = \frac{F_x}{A_x} = 7.31 \text{ ksi}\)

Ratio = 0.497

**Check Against Bending**

Allowable Stress along Major Axis (Z-Axis):

Since as per clause NF 3322.1(d)(1)(a)(7),

\[ \frac{OD}{t_w} = 12.75/0.35 = 36.429 < 3,300/F_y = 91.67 \]

Thus, allowable bending stress is obtained by treating the section as compact.

\[ F_{bcz} = 0.66 \times F_y = 23.76 \text{ ksi} = F_{btz} \]

Allowable Stress along Minor Axis (Y-Axis):

Here also, allowable stress along minor axis is given as \(F_{by} = 0.66 \times FYLD = 23.8 \text{ ksi} = F_{bcy}\)

Actual Bending Stress along Major Axis (Z-Axis):

\[ M_z = 97.683 \text{ in} \cdot \text{k} \]

Bending Tensile Stress \(f_{btz} = (Mz/Iz) \times (OD/2) = 2.38 \text{ ksi} < 23.8 \text{ ksi}, \text{Hence OK.}\)

Bending Compressive Stress \(f_{bcz} = 2.38 \text{ ksi} < 23.8 \text{ ksi}, \text{Hence safe.}\)

Actual Bending Stress along Minor Axis (Y-Axis):

\[ M_y = 10.798 \text{ in} \cdot \text{k} \]

Bending Tensile Stress \(f_{by} = (Mz/Iz) \times (OD/2) = 0.263 \text{ ksi} < 23.8 \text{ ksi}, \text{Hence OK.}\)

Bending Compressive Stress \(f_{bcy} = 0.263 \text{ ksi} < 23.8 \text{ ksi}, \text{Hence safe.}\)

**Check for Combined Axial Compression and Bending**

\(C_{mz} = C_{my} = 1 \text{ (as provided in the input)}\)

Now, \(f_a = 7.31 \text{ ksi, } F_a = 14.7 \text{ ksi, } F_{bcz} = 2.38 \text{ ksi, } F_{bcy} = 23.8 \text{ ksi, } f_{bcy} = 0.263 \text{ ksi, } F_{bcy} = 23.8 \text{ ksi}\)

Since \(f_a / F_a = 7.31 / 14.7 = 0.497 > 0.15\)
\[ F_{ez}' = \frac{n^2 \times E}{2.15 \times \left( \frac{kl}{r} \right)^2 z} = 176.804 \text{ ksi} \]  
\[ (\text{NF 3322.1(e)(1)} ) \]

\[ F_{ey}' = \frac{n^2 \times E}{2.15 \times \left( \frac{kl}{r} \right)^2 y} = 176.804 \text{ ksi} \]  
\[ (\text{NF 3322.1(e)(1)} ) \]

From Eqn. 20:
\[
\frac{f_a}{F_{a}} + \frac{C_{mz} \times f_{bz}}{F_{ez}} \times F_{bz} + \frac{C_{my} \times F_{by}}{F_{ey}} \times F_{by} = 0.613 < 1, \text{ Hence safe (critical ratio)}
\]

From Eqn. 21:
\[
\frac{f_a}{0.6F_{y}} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = 0.449 < 1, \text{ Hence safe}
\]

**Check Against Shear**

Actual Shear along Major Axis:
\[
V_y = 2 \times \text{OD} \times t_w = 8.925 \text{ in}^2
\]
\[
f_{vy} = 5 \text{ kip} / 8.925 \text{ in}^2 = 0.56 \text{ ksi}
\]
(No shear along minor axis; \( f_{vz} = 0 \))

Allowable Shear:
As per NV 3322.1(b)(1), allowable shear stress, \( F_v = 0.4 \times F_y = 14.4 \text{ ksi} \)

Ratio = 0.039

**Results**

**Table 632: Comparison of results**

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slenderness ratio</td>
<td>27.44</td>
<td>27.44</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual compressive stress (ksi)</td>
<td>7.31</td>
<td>7.31</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable compressive stress (ksi)</td>
<td>14.7</td>
<td>14.7</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Actual bending stress; major axis (ksi)</td>
<td>2.38</td>
<td>2.38</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress, major axis (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
### Result Type

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actual bending stress, minor axis (ksi)</td>
<td>0.263</td>
<td>0.263</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Allowable bending stress, minor axis (ksi)</td>
<td>23.8</td>
<td>23.8</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Combined interaction ratio</td>
<td>0.613</td>
<td>0.613</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Shear stress, major axis (ksi)</td>
<td>0.039</td>
<td>0.039</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD.Pro Input File

The file `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\NF 3000-2004\ASME NF 3000 2004 STYPE 1 Pipe.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Mar-19
END JOB INFORMATION
*****************************************************************************
*This problem has been created to validate the program calculated design
*results of ASME NF 3000 2004 pipe section with STYPE 1 parameter
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 10 0; 6 0 10 10; 7 10 10 10;
8 10 0 10;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 8 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 5 8 FIXED
LOAD 1 LOADTYPE Dead TITLE DL-1
```
Joint Load
6 FX 100
7 FX -100
Load 2 Loadtype Dead Title DL-2
Member Load
2 4 5 7 Uni GY -1
Load Comb 3 Combination Load Case 3
1 1.0 2 1.0
Perform Analysis
Unit Inches Kip
Print Member Forces List 7
Section 0.25 0.5 0.75
Print Member Section Forces List 7
Print Material Properties List 7
Print Member Properties List 7
Load List 3
Parameter 1
Code NF3000 2004
Stype 1 All
Cmy 1 All
Track 2 All
Check Code Memb 7
Finish

STAAD.Pro Output

1. STAAD SPACE
Input File: ASME NF 3000 2004 Stype 1 Pipe.STD
2. Start Job Information
3. Engineer Date 20-MAR-19
4. End Job Information

7. *This Problem Has Been Created To Validate The Program Calculated Design
8. *Results Of ASME NF 3000 2004 Pipe Section With Stype 1 Parameter

11. Input Width 79
12. Set Shear
13. Unit Feet Kip
14. Joint Coordinates
15. 1 0 0 0; 2 10 0 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10
### Verification Examples

#### V.09 Steel Design

---

16. 8 10 0 10
17. MEMBER INCIDENTS
18. 1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8
19. DEFINE MATERIAL START
20. ISOTROPIC STEEL
21. E 4.176E+06
22. POISSON 0.3
23. DENSITY 0.489024
24. ALPHA 6.5E-06
25. DAMP 0.03
26. TYPE STEEL
27. STRENGTH RY 1.5 RT 1.2
28. END DEFINE MATERIAL
29. MEMBER PROPERTY AMERICAN
30. 1 TO 8 TABLE ST PIPS120
31. CONSTANTS
32. MATERIAL STEEL ALL
33. SUPPORTS
34. 1 4 5 8 FIXED
35. LOAD 1 LOADTYPE DEAD TITLE DL-1
36. JOINT LOAD
37. 6 FX 100
38. 7 FX -100
39. LOAD 2 LOADTYPE DEAD TITLE DL-2
40. MEMBER LOAD
41. STAAD SPACE

---

### Problem Statistics

---

**NUMBER OF JOINTS**: 8  **NUMBER OF MEMBERS**: 8
**NUMBER OF PLATES**: 0  **NUMBER OF SOLIDS**: 0
**NUMBER OF SURFACES**: 0  **NUMBER OF SUPPORTS**: 4

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 24
TOTAL LOAD COMBINATION CASES = 1 SO FAR.

---

45. UNIT INCHES KIP
46. PRINT MEMBER FORCES LIST 7
MEMBER FORCES LIST 7
STAAD SPACE

---

### Member End Forces

---

<table>
<thead>
<tr>
<th>MEMBER LOAD JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>7 1</td>
<td>6</td>
<td>99.31</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-10.80</td>
</tr>
<tr>
<td>7 -99.31</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-10.80</td>
<td>14.22</td>
</tr>
<tr>
<td>2 6</td>
<td>0.83</td>
<td>5.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>66.53</td>
</tr>
<tr>
<td>7 -0.83</td>
<td>5.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>66.53</td>
</tr>
<tr>
<td>3 6</td>
<td>100.14</td>
<td>5.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>10.80</td>
<td>52.32</td>
</tr>
<tr>
<td>7 -100.14</td>
<td>5.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-10.80</td>
<td>-52.32</td>
<td></td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

47. SECTION 0.25 0.5 0.75
### Member Forces at Intermediate Sections

All units are in KIP inches.

<table>
<thead>
<tr>
<th>Mem</th>
<th>Load</th>
<th>Sec</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Mom-X</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>1</td>
<td>0.25</td>
<td>99.31</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>10.80</td>
</tr>
<tr>
<td></td>
<td>-14.22</td>
<td>0.50</td>
<td>99.31</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>10.80</td>
</tr>
<tr>
<td></td>
<td>-14.22</td>
<td>0.75</td>
<td>99.31</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>10.80</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.25</td>
<td>0.83</td>
<td>2.50</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>-45.97</td>
<td>0.50</td>
<td>0.83</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td>-83.47</td>
<td>0.75</td>
<td>0.83</td>
<td>-2.50</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td>-45.97</td>
<td>3</td>
<td>100.14</td>
<td>2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>10.80</td>
</tr>
<tr>
<td></td>
<td>-60.18</td>
<td>0.50</td>
<td>100.14</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>10.80</td>
</tr>
<tr>
<td></td>
<td>-97.68</td>
<td>0.75</td>
<td>100.14</td>
<td>-2.50</td>
<td>-0.00</td>
<td>0.00</td>
<td>10.80</td>
</tr>
</tbody>
</table>

*********** END OF LATEST ANALYSIS RESULT ***********

### Material Properties

All units are in KIP inches.

<table>
<thead>
<tr>
<th>Material</th>
<th>Kind</th>
<th>E</th>
<th>Pois</th>
<th>Dens</th>
<th>Alpha</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>1D</td>
<td>2.90000E+04</td>
<td>0.300</td>
<td>2.83000E-04</td>
<td>6.50000E-06</td>
</tr>
</tbody>
</table>

0.030 1.11538E+04

### Member Properties

<table>
<thead>
<tr>
<th>Mem</th>
<th>Profile</th>
<th>Ax/</th>
<th>Iz/</th>
<th>Iy/</th>
<th>IX/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>St</td>
<td>AY</td>
<td>AZ</td>
<td>SZ</td>
<td>SY</td>
</tr>
<tr>
<td>7</td>
<td>PIPS120</td>
<td>13.70</td>
<td>262.00</td>
<td>262.00</td>
<td>524.00</td>
</tr>
</tbody>
</table>
### Steel Design

**STAAD SPACE**

-- PAGE NO.

8

**STAAD.PRO CODE CHECKING - (ASME NF3000-04) v2.0**

<table>
<thead>
<tr>
<th>ALL UNITS ARE - KIP INCH (UNLESS OTHERWISE Noted)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/CRITICAL COND/LOADING/LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
</tr>
<tr>
<td>PASS</td>
<td>NF-EQN-20</td>
<td>0.613</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100.14 C</td>
</tr>
<tr>
<td></td>
<td></td>
<td>-97.68</td>
</tr>
</tbody>
</table>

---

**Slenderness Check:**

- **Actual Ratio:** 27.44
- **Allowable Ratio:** 200.00

**Allowable Stresses:**

- **Axial:** 1.47E+01
- **FCZ:** 2.38E+01
- **FCY:** 2.38E+01
- **FTZ:** 2.38E+01
- **FTY:** 2.38E+01

**Shear:** 1.44E+01

**Actual Stresses:**

- **Axial:** 7.31E+00
- **FBZ:** 2.38E+00
- **FBY:** 2.63E-01
- **Shear:** 0.00E+00

---

**Section Properties:**

- **A**:
  - **AXX:** 13.70
  - **AYY:** 7.01
  - **AZZ:** 7.01
  - **AZZ:** 4.37
  - **RYY:**
  - **SZZ:** 41.10
  - **SYY:**
  - **41.10**

---

**Parameter:**

- **KL/R-Z:** 27.44
- **KL/R-Y:** 27.44
- **UNL:** 120.0
- **CMZ:** 1.00
- **CMY:**
- **CB:** 1.00
- **FYLD:** 36.00
- **FU:** 58.02
- **Net Section Factor:**
- **CT:** 0.75
- **Steel Type:**
  - **KS:** 1.00
  - **KV:** 1.00
  - **KBK:**

---

**Verification Examples**

V.09 Steel Design
### CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP - INCH)

<table>
<thead>
<tr>
<th>CLAUSE</th>
<th>RATIO</th>
<th>LOAD</th>
<th>FX</th>
<th>VY</th>
<th>VZ</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>TENSION</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMPRESSION</td>
<td>0.497</td>
<td>3</td>
<td>1.00E+02</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>COMP&amp;BEND</td>
<td>0.613</td>
<td>3</td>
<td>1.00E+02</td>
<td>-</td>
<td>-</td>
<td>9.77E+01 1.08E</td>
</tr>
<tr>
<td>TEN&amp;BEND</td>
<td>0.000</td>
<td>0</td>
<td>0.00E+00</td>
<td>-</td>
<td>-</td>
<td>0.00E+00 0.00E</td>
</tr>
<tr>
<td>SHEAR-Y</td>
<td>0.039</td>
<td>3</td>
<td>-</td>
<td>5.00E+00</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>SHEAR-Z</td>
<td>0.000</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>0.00E+00</td>
<td>-</td>
</tr>
</tbody>
</table>

---

**59. FINISH**

************** END OF THE STAAD.Pro RUN **************

**** DATE= APR 14,2019 TIME= 23:21:41 ****

STAAD SPACE                                              -- PAGE NO. 9

**********************************************
* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/ *
* *
* Details about additional assistance from *
* Bentley and Partners can be found at program menu *
* Help->Technical Support *
* *
* Copyright (c) 1997-2017 Bentley Systems, Inc. *
* http://www.bentley.com *
**********************************************

---

### V.10 Concrete Design

### V. United States

### V. ACI 318-1999

### V. ACI 318-99 Square Column

Determine the reinforced steel quantity for the square column as per ACI 318-99.
Reference


Problem

\[ P_u = 144 \text{ kips} \]
\[ M_{uz} = 120 \text{ ft·kips} \]
\[ M_{uy} = 54 \text{ ft·kips} \]
\[ f'_c = 3,000 \text{ psi} \]
\[ f_y = 60,000 \text{ psi} \]

Comparison

Table 633: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Provided steel area per ACI 318-99(in²)</td>
<td>8.0 (8- #9)</td>
<td>9.0 (4- #14)</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: In the reference book, the axial capacity provided is 141 kips, which is less than the requirement of 144 kips. In the STAAD implementation of the code, any reinforcement arrangement which yields less capacity than the required value is considered unacceptable. This is why STAAD reports a higher required steel than the reference book.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-1999\ACI 318-99 Square Column.STD is typically installed with the program.

START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION

* REFERENCE :  REINFORCED CONCRETE DESIGN 5TH EDITION BY CHU-KIA WANG

* OBJECTIVE : TO DETERMINE THE REINFORCED STEEL QUANTITY FOR A
SQUARE COLUMN PER THE ACI 318-99 CODE

UNIT FEET KIP
JOIN COORDINATES
1 0 0 0; 2 0 12 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 PRIS YD 16 ZD 16
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
UNIT FEET KIP
LOAD 1
JOINT LOAD
2 FY -144
2 MZ 120
2 MX 54
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 1999
UNIT INCHES KIP
FC 3 ALL
FYMAIN 40 ALL
FYSEC 40 ALL
MINMAIN 9 ALL
TRACK 2 ALL
DESIGN COLUMN ALL
END CONCRETE DESIGN
FINISH

STAAD Output

====================================================================
COLUMN NO. 1 DESIGN PER ACI 318-99 - AXIAL + BENDING
FY - 40000 FC - 3000 PSI, SQRE SIZE - 16.00 X 16.00 INCHES, TIED
AREA OF STEEL REQUIRED = 8.348 SQ. IN.
BAR CONFIGURATION REINF PCT. LOAD LOCATION PHI
----------------------------------------------------------
4 - NUMBER 14 3.516 1 END 0.700
(PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
TIE BAR NUMBER 4 SPACING 16.00 IN
COLUMN INTERACTION: MOMENT ABOUT Z-AXIS (KIP-FT)
--------------------------------------------------------
P0  Pn max P-bal. M-bal. e-bal.(inch)
989.85 791.88 312.89 272.99 10.47
M0  P-tens. Des.Pn Des.Mn e/h
178.89 -360.00 205.71 171.43 0.06944
--------------------------------------------------------
COLUMN INTERACTION: MOMENT ABOUT Y-AXIS (KIP-FT)
--------------------------------------------------------
P0  Pn max P-bal. M-bal. e-bal.(inch)
989.85 791.88 312.89 272.99 10.47
M0  P-tens. Des.Pn Des.Mn e/h
178.89 -360.00 205.71 77.14 0.03125
--------------------------------------------------------
Pn Mn Pn Mn (@ Z )
730.97 130.65 365.48 257.21
670.85 156.16 304.57 272.89
609.14 178.25 243.66 267.81
548.22 199.53 182.74 255.16

Verification Examples
V.10 Concrete Design
STAAD.Pro 4252 User Manual
V. ACI 318-99 Circular Column

Determine the reinforced steel quantity for a circular column per ACI 318-99.

Reference


Problem

\[ P_u = 1,050 \text{ kips} \]
\[ M_{uz} = M_{uy} = 0 \text{ ft-kips} \]
\[ f'_c = 4,000 \text{ psi} \]
\[ f_y = 60,000 \text{ psi} \]

Spiral lateral reinforcement used.

Comparison

Table 634: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required steel area (in(^2))</td>
<td>10.45</td>
<td>10.505</td>
<td>none</td>
</tr>
<tr>
<td>Provided steel area (in(^2))</td>
<td>10.82 (7- #11)</td>
<td>10.82 (18- #7)</td>
<td>none</td>
</tr>
</tbody>
</table>

Note: STAAD reports a more economical bar arrangement than the reference book.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-1999\ACI 318-99 Circular Column.STD is typically installed with the program.

STAAD SPACE CIRCULAR CONCRETE COLUMN DESIGN PER ACI 318-99
START JOB INFORMATION
V. ACI 318-99 Beam and Column Reinforcement

Determine the area of reinforcing steel required for a beam and column in a moment frame.
Reference

Problem
A plane frame is created with such loading as to create 138 Kip-Ft moment on beam and 574 Kip of axial load coupled with above moment on column.

Size of the beam is 10” x 16” and size of the column 14” x 16”.

\[
\begin{align*}
P &= 521.32 \text{ kip} \\
w &= 5.268 \text{ kip/ft} \\
L &= 20 \text{ ft}, H = 15 \text{ ft}
\end{align*}
\]

![Concrete moment frame](image)

*Figure 454: Concrete moment frame*

Comparison

Table 635: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Theory ACI Notes</th>
<th>Theory CRSI Handbook</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area of Steel in beam</td>
<td>2.78 in²</td>
<td>X</td>
<td>2.792 in²</td>
<td>negligible</td>
</tr>
<tr>
<td>Area of Steel in column</td>
<td>X</td>
<td>4.01%</td>
<td>4.09% required(*) 4.232% provided</td>
<td>negligible</td>
</tr>
</tbody>
</table>

*Note*: (*) Area of steel required calculated as 9.164 in², which is 4.09% of a 14”x16” column.
STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-1999\ACI 318-99 Beam and Column Reinforcement.STD is typically installed with the program.

STAAD PLANE VERIFICATION FOR CONCRETE DESIGN
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 15 0; 3 20 15 0; 4 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 3 PRIS YD 16 ZD 14
2 PRIS YD 16 ZD 10
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 FIXED
UNIT FEET KIP
LOAD 1 DEAD + LIVE
JOINT LOAD
2 3 FY -521.32
MEMBER LOAD
2 UNI GY -5.268
PERFORM ANALYSIS
PRINT MEMBER FORCES
START CONCRETE DESIGN
CODE ACI 1999
UNIT INCHES KIP
TRACK 1 MEMB 2
FYMAIN 60 ALL
FC 4 ALL
CLB 1.4375 ALL
DESIGN BEAM 2
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH

STAAD Output

V. ACI 318-2002
V. ACI 318-02 Rectangular Beam

Determine the reinforced steel quantity for a rectangular beam per the ACI 318 code.

Reference

Notes on ACI 318-02 Building Code Requirements for Structural Concrete, Example 7.1, p7-24, Design of Rectangular Beam with Tension Reinforcement Only

Notes on ACI 318-99 Building Code Requirements for Structural Concrete, Example 10.1, p10-10, Design of Rectangular Beam with Tension Reinforcement Only

Problem

Dead Load moment (service) = 56 ft-kips
Live Load moment (service) = 35 ft-kips
$f_c' = 4,000$ psi
$f_y = 60,000$ psi

Comparison

Table 636: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>ACI Notes</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required steel area</td>
<td>per ACI 318-99</td>
<td>2.78</td>
<td>2.793</td>
</tr>
<tr>
<td>(in²)</td>
<td>per ACI 318-02</td>
<td>2.40</td>
<td>2.44</td>
</tr>
<tr>
<td>Provided steel area</td>
<td>per ACI 318-02</td>
<td>2.40 (2 - #8 &amp; 1 - #9)</td>
<td>(2 - #10) 2.45</td>
</tr>
<tr>
<td>(in²)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

STAAD.Pro reports that it is unable to find a suitable bar arrangement to satisfy the reinforcement requirement per ACI 318-99. However, this does not mean that it is impossible to come up with a bar arrangement. When STAAD looks for a bar arrangement, it uses only bars of the same size. It begins with the bar size corresponding to the parameter \textit{MINMAIN}. If an arrangement is not possible with that bar, it tries with the next larger bar size. If all the permissible bar sizes are exhausted, the program reports that it could not come up with a bar arrangement. However, the user may be able to satisfy the requirement by mixing bars of various diameters. For example, 3 # 11 bars and 2 # 10 bars may satisfy the requirement. The program is not equipped with facilities to come up with such combinations of bar sizes.

In fact, in the reference mentioned for the 1999 edition of ACI 318, the bar arrangement chosen is 2 # 9 and 1 # 8. It proves the point that finding bars of the same diameter is not possible for the 1999 edition based solution.

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-2002\ACI 318-02 Rectangular Beam.STD is typically installed with the program.

START JOB INFORMATION
**REFERENCES:**

- Notes on ACI 318-02 Building Code Requirements for Structural Concrete Example 7.1 (Page # 7-24)
- Notes on ACI 318-99 Building Code Requirements for Structural Concrete Example 10.1 (Page # 10-10)

**OBJECTIVE:**

To determine the reinforced steel quantity for a rectangular beam per the ACI 318 Code.

**Input Width 79**

**Joint Coordinates**

1 0 0 0; 2 10 0 0;

**Member Incidences**

1 1 2;

**Unit Inches KIP**

**Member Property American**

1 PRIS YD 16 ZD 10

**Define Material Start**

**Isotropic Material1**

E 3150

POISSON 0.17

**End Define Material**

**Constants**

**Material Material1 All**

**Supports**

1 2 PINNED

**Unit Feet KIP**

**Load 1 Dead Load**

MEMBER LOAD 1 UNI GY -4.48

**Load 2 Live Load**

MEMBER LOAD 1 UNI GY -2.8

**Load Comb 3 ACI 318R-99**

1 1.4 2 1.7

**Load Comb 4 ACI 318R-02**

1 1.2 2 1.6

**Perform Analysis**

**Concrete Design as Per ACI 318R-99**

**Concrete Design as Per ACI 318R-02**
LOAD LIST 4
START CONCRETE DESIGN
CODE ACI 2002
FC 4 ALL
FYMAIN 60 ALL
TRACK 1 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH

STAAD Output

RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318 -- PAGE NO. 3
=====================================================================
BEAM NO. 1 DESIGN RESULTS - FLEXURE PER CODE ACI 318-99
LEN - 10.00FT. FY - 60000. FC - 4000. SIZE - 10.00 X 16.00 INCHES
LEVEL HEIGHT BAR INFO FROM TO ANCHOR
FT. IN. FT. IN. FT. IN. STA END

*** A SUITABLE BAR ARRANGEMENT COULD NOT BE DETERMINED.
REQD. STEEL = 2.793 IN2, MAX. STEEL PERMISSIBLE = 2.873 IN2
MAX POS MOMENT = 137.90 KIP-FT, LOADING 3

1 120.X 10.X 16_____________________ 2J____

***************END OF BEAM DESIGN***************
57. END CONCRETE DESIGN
58. *
59. ** CONCRETE DESIGN AS PER ACI 318R-02
60. *
61. LOAD LIST 4
62. START CONCRETE DESIGN

RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318 -- PAGE NO. 4
=====================================================================
### BEAM NO. 1 DESIGN RESULTS - FLEXURE PER CODE ACI 318-02

<table>
<thead>
<tr>
<th>LEVEL</th>
<th>HEIGHT</th>
<th>BAR INFO</th>
<th>FROM</th>
<th>TO</th>
<th>ANCHOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>FT.</td>
<td>IN.</td>
<td>FT.</td>
<td>IN.</td>
<td>FT.</td>
<td>IN.</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>2-3/4</td>
<td>0</td>
<td>0</td>
<td>0/0</td>
</tr>
</tbody>
</table>

**Critical Pos Moment:** 123.20 KIP-FT at 5.00 FT, LOAD 4

**Reqd Steel:** 2.44 IN², RHO=0.0183, RHOMX=0.0214, RHOMN=0.0033

**Max/Min/Actual Bar Spacing:** 10.00/2.54/4.73 INCH

**Reqd. Development Length:** 48.52 INCH

Cracked Moment of Inertia Iz at above location = 1817.34 inch^4

---

### BEAM NO. 1 DESIGN RESULTS - SHEAR

**At Start Support:**
- Vu= 38.19 KIP
- Vc= 21.42 KIP
- Vs= 29.50 KIP
- Tu= 0.00 KIP-FT
- Tc= 1.95 KIP-FT
- Ts= 0.00 KIP-FT

Load 4

**No Stirrups Are Required for Torsion.**

**Reinforcement Is Required for Shear.**

Provide Num. 4 2-Legged Stirrups at 6.7 IN. C/C for 46. IN.

**At End Support:**
- Vu= 38.19 KIP
- Vc= 21.42 KIP
- Vs= 29.50 KIP
- Tu= 0.00 KIP-FT
- Tc= 1.95 KIP-FT
- Ts= 0.00 KIP-FT

Load 4

**No Stirrups Are Required for Torsion.**

**Reinforcement Is Required for Shear.**

Provide Num. 4 2-Legged Stirrups at 6.7 IN. C/C for 46. IN.

**Rectangular Concrete Beam Design Per ACI 318**

---

**V. ACI 318-02 Square Column**

Determine the reinforced steel quantity for a square column per ACI 318-02.

**Reference**

*Notes on ACI 318-02 Building Code Requirements for Structural Concrete, Example 7.8, p7-46, Design of Square Column for Biaxial loading.*
Problem

\[ P = 1,200 \text{ kips} \]
\[ M_{uz} = 300 \text{ ft·kips} \]
\[ M_{uy} = 125 \text{ ft·kips} \]
\[ f'_{c} = 5,000 \text{ psi} \]
\[ f_{y} = 60,000 \text{ psi} \]

Comparison

Table 637: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>ACI Notes</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Provided steel area per ACI 318-02 (in²)</td>
<td>5.93 (4- #11)</td>
<td>5.93 (4- #11)</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-2002\ACI 318-02 Square Column.STD is typically installed with the program.

**STAAD SPACE SQUARE COLUMN DESIGN PER ACI 318-02**

**START JOB INFORMATION**

**ENGINEER DATE 21-Sep-18**

**END JOB INFORMATION**

*REFERENCE : NOTES ON ACI 318-02 BUILDING CODE REQUIREMENTS*

*FOR STRUCTURAL CONCRETE EXAMPLE 7.8 (PAGE # 7-46)*

*OBJECTIVE : TO DETERMINE THE REINFORCED STEEL QUANTITY FOR A SQUARE COLUMN PER THE ACI 318-02 CODE*

**UNIT FEET KIP**

**JOINT COORDINATES**

1 0 0 0; 2 0 12 0;

**MEMBER INCIDENCES**

1 1 2;

**UNIT INCHES KIP**

**MEMBER PROPERTY AMERICAN**

1 PRIS YD 24 ZD 24

**DEFINE MATERIAL START**

**ISOTROPIC MATERIAL1**

E 3150

**POISSON 0.17**

**END DEFINE MATERIAL**

**CONSTANTS**

**MATERIAL MATERIAL1 ALL SUPPORTS**

1 FIXED

**UNIT FEET KIP**

**LOAD 1**
STAAAD Output

SQUARE COLUMN DESIGN PER ACI 318-02

PROBLEM STATISTICS

NUMBER OF JOINTS 2
NUMBER OF MEMBERS 1
NUMBER OF PLATES 0
NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0
NUMBER OF SUPPORTS 1

Using 64-bit analysis engine.

SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 6
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

CONCRETE DESIGN

CODE ACI 2002
UNIT INCHES KIP
FC 5 ALL
MINMAIN 11 ALL
MAXMAIN 11 ALL
TRACK 2 ALL
DESIGN COLUMN ALL

SQUARE COLUMN DESIGN PER ACI 318-02

COLUMN NO. 1 DESIGN PER ACI 318-02 - AXIAL + BENDING
FY - 60000  FC - 5000 PSI,  SQRE SIZE - 24.00 X 24.00 INCHES, TIED
ONLY MINIMUM STEEL IS REQUIRED.
AREA OF STEEL REQUIRED = 5.760  SQ. IN.

BAR CONFIGURATION  REINF PCT. LOAD LOCATION PHI

4 - NUMBER 11 1.083 1 END 0.650

(TIE BAR NUMBER 4 SPACING 22.56 IN)

COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)

P0  Pn max  P-bal.  M-bal.  e-bal.(inch)
2795.88  2236.70  1039.31  894.77  10.33
M0  P-tens.  Des.Pn  Des.Mn  e/h
325.72  -374.40  1846.15  461.54  0.02083

Verification Examples
V.10 Concrete Design

STAAD.Pro 4262 User Manual
V. ACI 318-2014

V. ACI 318-14 Circular Column

Calculate the axial capacity of a circular column subjected to axial compressive force, given $A_{st}$

**Details**

A circular, cantilever column of length 5ft with an ultimate axial load of 500 kip has the following concrete and steel properties.

- Compressive strength of concrete (normal weight concrete), $f_c = 4,000$ psi
- Yield strength of steel, $f_y = 60,000$ psi
- Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5$ in
- Diameter of column, $D = 20$ in
- Strength reduction factor, $\phi = 0.65$
- Number of #8 bars = 6
- Rebar diameter for #8 bar, $\varnothing d = 1$ in

Factored axial load (axial demand), $P_u = 500$ kip

**Validation**

Gross area of concrete section, $A_g = \pi D^2/4 = 314.159$ in$^2$
Total area of steel reinforcement, \( A_{st} = \left( \pi \frac{D^2}{4} \right) N = 314.159 \text{in}^2 \)

Nominal axial strength at zero eccentricity:

\[
P_0 = 0.85 \times f_c (A_g - A_{st}) + f_y A_{st} = 1.335 \times 10^3 \text{kip}
\]

For tied reinforced column:

\[
P_{n(max)} = 0.8 \times P_0 = 1.068 \times 10^3 \text{kip}
\]

Capacity = \( \phi P_{n(max)} = 694.129 \text{kip} \)

Ratio = \( P_u / \phi P_{n(max)} = 0.72 \)

**Results**

**Table 638: Comparison of results**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Capacity (kip)</td>
<td>694.129</td>
<td>694.079</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Ratio ( P_u / \phi P_{n(max)} )</td>
<td>0.72</td>
<td>0.72</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-2014\ACI 318-14 Circular Column.STD is typically installed with the program.

**STAAD SPACE**

START JOB INFORMATION
ENGINEER DATE 29-Aug-17
JOB NAME Circular Column Design
JOB CLIENT Bentley Systems Inc.
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 1.66667
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
JOINT LOAD
2 FY -500
PERFORM ANALYSIS
PRINT SUPPORT REACTION ALL
START CONCRETE DESIGN
CODE ACI 318 14
MAXMAIN 9 ALL
MINMAIN 8 ALL
TRACK 2 ALL
DESIGN COLUMN ALL
END CONCRETE DESIGN
FINISH

**STAAD Output**

**STAAD.PRO CONCRETE DESIGN - (ACI-318-14)**

v2.0

*****************

Units: KIP, FEET (Unless Noted Otherwise)

Member : 1

**DESIGN SUMMARY**

| Status         : Pass         | Type    : Column          |
| Critical Ratio : 0.720       | Criteria: Axial            |
| Critical Clause: 10.5.2       |

| Shape: Circular | Dia: 1.67 |

**DESIGN INPUTS**

| Concrete | Fc | 576.000 |
| Steel    | Fy(main) | 8639.999 |
| Cover    | Top | 0.125   |
|          | Bottom | 0.125   |
|          | Sides | 0.125   |

**CRITICAL STRENGTH RESULTS**

<table>
<thead>
<tr>
<th>Category</th>
<th>Demand</th>
<th>Min Capacity</th>
<th>Max Capacity</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>-500.000</td>
<td>-694.079</td>
<td>255.960</td>
<td>0.720</td>
</tr>
<tr>
<td>Flexure</td>
<td>0.000</td>
<td>-148.011</td>
<td>148.011</td>
<td>0.000</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Shear Z</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Torsion</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

**LONGITUDINAL BAR DETAILS AT CROSS SECTIONS**

<table>
<thead>
<tr>
<th>Distance</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>0.022</td>
<td>0.033</td>
<td>6</td>
<td># 8</td>
<td>1</td>
</tr>
<tr>
<td>1.250</td>
<td>0.022</td>
<td>0.033</td>
<td>6</td>
<td># 8</td>
<td>1</td>
</tr>
<tr>
<td>2.500</td>
<td>0.022</td>
<td>0.033</td>
<td>6</td>
<td># 8</td>
<td>1</td>
</tr>
</tbody>
</table>
Related Links

- **D1.F.5 Column Design** (on page 1509)

**V. ACI 318-14 Rectangular Column**

Calculate the axial capacity of a rectangular column subjected to axial compressive force, given $A_t$

**Details**

A rectangular, cantilever column of length 5ft with an ultimate axial load of 1,538 kip has the following concrete and steel properties.

- Compressive strength of concrete (normal weight concrete), $f_c = 5,000 \text{ psi}$
- Yield strength of steel, $f_y = 60,000 \text{ psi}$
- Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5 \text{ in}$
- Width of column, $B = 20 \text{ in}$
- Depth of column, $D = 30 \text{ in}$
- Strength reduction factor, $\phi = 0.65$
- Number of #9 bars = 10
- Rebar diameter for #9 bar, $\phi_d = 1.128 \text{ in}$

Factored axial load (axial demand), $P_u = 1,538 \text{ kip}$
**Validation**

Gross area of concrete section, \( A_g = B \times D = 600 \text{ in}^2 \)

Total area of steel reinforcement, \( A_{st} = 10 \text{ in}^2 \)

Nominal axial strength at zero eccentricity:

\[
P_0 = 0.85f_c(A_g - A_{st}) + fyA_{st} = 3.108 \times 10^3 \text{ kip}
\]

For tied reinforced column:

\[
P_{n(max)} = 0.8P_0 = 2.486 \times 10^3 \text{ kip}
\]

Capacity = \( \phi P_{n(max)} = 1,615.9 \text{ kip} \)

Ratio = \( P_u/\phi P_{n(max)} = 0.952 \)

**Results**

Table 639: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Capacity (kip)</td>
<td>1,615.9</td>
<td>1,615.9</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Ratio ( P_u/\phi P_{n(max)} )</td>
<td>0.952</td>
<td>0.952</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-2014\ACI 318-14 Rectangular Column.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 01-Jan-18
JOB CLIENT Bentley Systems Inc.
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-05
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 30 ZD 20
**Verification Examples**

**V.10 Concrete Design**

---

```plaintext
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET KIP
SUPPORTS
1 FIXED
UNIT INCHES KIP
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FY -1538
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 2014
FC 5 ALL
MXMT 9 ALL
MXMB 9 ALL
MXMS 9 ALL
MIMT 9 ALL
MIMB 9 ALL
MIMS 9 ALL
TRACK 2 ALL
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

---

**STAAD Output**

```
STAAD.PRO CONCRETE DESIGN - (ACI-318-14)
v2.0

Units: KIP   , INCHES (Unless Noted Otherwise)
Member :      1
DESIGN SUMMARY

| Status         :   Pass         | Type    : Column          | Length:   72.000 |
| Critical Ratio :   0.952        | Criteria: Axial                            |
| Critical Clause:   10.5.2                                                  |

CROSS SECTION

| Shape: Rectangular | Width:   20.00 | Depth:   30.00 |

DESIGN INPUTS

<table>
<thead>
<tr>
<th>Concrete</th>
<th>Fc</th>
<th>Steel</th>
<th>Fy(main)</th>
<th>Fy(trans)</th>
<th>Cover</th>
<th>Top</th>
<th>Bottom</th>
<th>Sides</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fc</td>
<td>5.000</td>
<td>Fy(main)</td>
<td>60.000</td>
<td>60.000</td>
<td>Cover</td>
<td>Top</td>
<td>Bottom</td>
<td>Sides</td>
</tr>
<tr>
<td>Steel</td>
<td>Fy(main)</td>
<td>Fy(trans)</td>
<td>60.000</td>
<td>60.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

CRITICAL STRENGTH RESULTS

<table>
<thead>
<tr>
<th>Category</th>
<th>Demand</th>
<th>Min Capacity</th>
<th>Max Capacity</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>-1538.000</td>
<td>-1615.900</td>
<td>540.000</td>
<td>0.952</td>
</tr>
<tr>
<td>Flexure</td>
<td>0.000</td>
<td>-4780.603</td>
<td>4780.603</td>
<td>0.000</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Shear Z</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Torsion</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro**  
**4268**  
**User Manual**
```
### Longitudinal Bar Details at Cross Sections

<table>
<thead>
<tr>
<th>Distance</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>8.660</td>
<td>10.000</td>
<td>10</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>18.000</td>
<td>8.660</td>
<td>10.000</td>
<td>10</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>36.000</td>
<td>8.660</td>
<td>10.000</td>
<td>10</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>54.000</td>
<td>8.660</td>
<td>10.000</td>
<td>10</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>72.000</td>
<td>8.660</td>
<td>10.000</td>
<td>10</td>
<td># 9</td>
<td>1</td>
</tr>
</tbody>
</table>

### Longitudinal Bar Layout

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars</th>
<th>Location</th>
<th>Distance</th>
<th>Anchor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>3</td>
<td># 9</td>
<td>0.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>3</td>
<td># 9</td>
<td>0.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Left</td>
<td>2</td>
<td># 9</td>
<td>0.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Right</td>
<td>2</td>
<td># 9</td>
<td>0.00</td>
<td>Yes</td>
</tr>
</tbody>
</table>

### Transverse Bar Layout

<table>
<thead>
<tr>
<th>Specification</th>
<th>Asv</th>
<th>Rebar</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zone</td>
<td>Dir.</td>
<td>From</td>
</tr>
<tr>
<td>Legs</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>Z</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**Related Links**

- [D1.F.5 Column Design](on page 1509)

**V. ACI 318-14 Square Column**

Calculate the axial capacity of a square column subjected to an axial compressive force, given Ast Details

A square, cantilever column of length 5ft with an ultimate axial load of 998 kip has the following concrete and steel properties.

- Compressive strength of concrete (normal weight concrete), $f_c = 5,000$ psi
- Yield strength of steel, $f_y = 60,000$ psi
- Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5$ in
- Width of column, $B = 20$ in
Depth of column, \( D = 20 \) in
Strength reduction factor, \( \phi = 0.65 \)
Number of \#9 bars = 4
Rebar diameter for \#9 bar, \( \phi_d = 1.128 \) in

Factored axial load (axial demand), \( P_u = 998 \) kip

Validation

Gross area of concrete section, \( A_g = B \times D = 400 \) in²
Total area of steel reinforcement, \( A_{st} = 4 \) in²
Nominal axial strength at zero eccentricity:
\[
P_0 = 0.85 f_c (A_g - A_{st}) + f_y A_{st} = 1.923 \times 10^3 \text{ kip}
\]
For tied reinforced column:
\[
P_{n(max)} = 0.8 P_0 = 1.5384 \times 10^3 \text{ kip}
\]
Capacity = \( \phi P_{n(max)} = 999.96 \) kip
Ratio = \( P_u / \phi P_{n(max)} = 0.998 \)

Results

Table 640: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Capacity (kip)</td>
<td>999.96</td>
<td>999.96</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>Ratio ( P_u / \phi P_{n(max)} )</td>
<td>0.998</td>
<td>0.998</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-2014\ACI 318-14 Square Column.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Aug-18
JOB CLIENT Bentley Systems Inc.
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
STAAD Output

STAAD.PRO CONCRETE DESIGN - (ACI-318-14)
v2.0

************

Units: KIP, INCHES (Unless Noted Otherwise)
Member: 1

DESIGN SUMMARY

| Status         : Pass | Type : Column | Length: 72.000 |
| Critical Ratio : 0.998 | Criteria: Axial |
| Critical Clause: 10.5.2 |

CROSS SECTION

| Shape: Rectangular | Width: 20.00 | Depth: 20.00 |

DESIGN INPUTS

| Concrete | Fc      | 5.000 |
| Steel    | Fy(main)| 60.000 |
| Cover    | Top    | 1.500 |

| Steel     | Fy(trans)| 60.000 |
| Cover     | Bottom  | 1.500 |
| Cover     | Sides   | 1.500 |

EC 0.403E+04
Es 0.290E+05

CRITICAL STRENGTH RESULTS

Verification Examples
V.10 Concrete Design
### Category Demand Min Capacity Max Capacity Ratio
- Axial -998.000 -999.960 216.000 0.998
- Flexure 0.000 -1740.458 1740.458 0.000
- Shear Y 0.000 0.000 0.000 0.000
- Shear Z 0.000 0.000 0.000 0.000
- Torsion 0.000 0.000 0.000 0.000

### LONGITUDINAL BAR DETAILS AT CROSS SECTIONS

<table>
<thead>
<tr>
<th>Distance</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>4.000</td>
<td>4.000</td>
<td>4</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>18.000</td>
<td>4.000</td>
<td>4.000</td>
<td>4</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>36.000</td>
<td>4.000</td>
<td>4.000</td>
<td>4</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>54.000</td>
<td>4.000</td>
<td>4.000</td>
<td>4</td>
<td># 9</td>
<td>1</td>
</tr>
<tr>
<td>72.000</td>
<td>4.000</td>
<td>4.000</td>
<td>4</td>
<td># 9</td>
<td>1</td>
</tr>
</tbody>
</table>

### LONGITUDINAL BAR LAYOUT

### TRANSVERSE BAR LAYOUT

### Related Links
- [D1.F.5 Column Design](on page 1509)

### V. ACI 318-14 Tee Beam

Calculate the flexural capacity of a T-beam subjected to major axis moment, given Ast

### References
1. ACI 318-14: Sections 19.2.1, 20.2.2, 22.3, and 22.4; Tables 22.2.2.4.3, 22.4.2.1, 27.3.2.1
2. PCA Notes 2011: Tables 6.1, 7.1
A T-beam of length 5ft is subjected to a major axis moment has the following concrete and steel properties.

- Width of T flange, \( B_f = 30 \text{ in} \)
- Depth of T, \( D = 26 \text{ in} \)
- Thickness of flange, \( T_f = 4 \text{ in} \)
- Thickness of web, \( T_w = 10 \text{ in} \)
- Steel modulus of elasticity, \( E_s = 29,000 \text{ ksi} \)
- Ultimate strain, \( \varepsilon_u = 0.003 \)
- Yield strength of reinforcing steel, \( f_y = 60 \text{ ksi} \)
- Compressive strength of concrete (normal weight concrete), \( f_c = 4,000 \text{ psi} \)
- Clear spacing for outermost bottom reinforcement to bottom face, \( C_c = 1.5 \text{ in} \)
- Strength reduction factor, \( \phi = 0.9 \)
- Rebar diameter for #10 bar, \( \phi d = 1.27 \text{ in} \)

Validation

\[
d = D - C_c - \frac{\phi d}{2} = 23.865
\]
\[
d' = C_c + \frac{\phi d}{2} = 2.135
\]
\[
R_n = \frac{6 \times 400}{\phi \times 0.85 \times f_y \times B_f \times d^2} = 0.122
\]

From Table 7.1 of PCA (ref. 2):

\[
w = \frac{(0.1726 - 0.1648)}{(0.19 - 0.18)} \times (R_n - 0.1648) + 0.18 = 0.147
\]

\[
a = 1.18 \times w \times d = 4.138 > T_f , \text{ hence beam acts as a T}
\]

Compression strength of the flange:

\[
C_f = 0.85 \times f_c \times (B_f + T_w) \times T_f = 136
\]

\[
A_{sf} = C_f f_y = 2.267 \text{ in}^2
\]

this is the required area to equilibrate \( C_f \).

Flexural strength of the flange:

\[
M_{nw} = \frac{6 \times 400}{\phi} - M_{nf} = 4.001 \times 10^{-3} \text{ in} \cdot \text{k}
\]

To check whether this section is tension controlled, refer to PCA Table 7.1 (ref. 2):

\[
R_n = \frac{M_{nw}}{f_c \times T_w \times d^2} = 0.176
\]
\[
w = \frac{(0.1802 - 0.1726)}{(0.2 - 0.19)} \times (R_n - 0.1726) + 0.19 = 0.192
\]
\[ \rho = w_n \frac{f_c}{f_y} = 0.013 < 0.01806 \]

based on PCA Table 6.1 (ref. 2), thus the section is tension controlled.

\[ A_{sw} = \rho \times T_w \times d = 3.06 \text{ in}^2 \]

Assume 6- #10 bars, divided between top and bottom: \( A_{st,tot} = 6 \times 1.27 \text{ in}^2 = 7.62 \text{ in}^2 \)

\[ a_w = \left( A_{st,tot} - A_{sf} \right) \frac{f_y}{0.85 \times f_c \times T_w} = 9.447 \text{in} \]

\[ M_n = (A_{st,tot} - A_{sf}) \times f_y \left( d - \frac{a_w}{2} \right) + A_{sf} \times f_y \times \left( d - \frac{T_f}{2} \right) = 9,258 \text{in}\cdot\text{k} \]

\[ \phi M_n = 8,332 \text{ in} \cdot \text{k} = 694.4 \text{ ft} \cdot \text{k} \]

Results

Table 641: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Hand Calculation</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexural capacity (ft·k)</td>
<td>694.4</td>
<td>698.938</td>
<td>&lt;1%</td>
<td>OK</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI\318-2014\ACI 318-14 Tee Beam.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 4 0; 2 5 4 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 2.16667 ZD 2.5 YB 1.83333 ZB 0.83333
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 MZ 533.3333
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI
MAXMAIN 10 ALL
MINMAIN 9 ALL
MXMS 5 ALL
MIMS 4 ALL
TRACK 2 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH

**STAAD Output**

STAAD.PRO CONCRETE DESIGN - (ACI-318-14)

v2.0

**********************************************
Units: KIP, FEET (Unless Noted Otherwise)

Member: 1

---

DESIGN SUMMARY

| Status         :   Pass         | Type    : Beam            | Length: 5.000 |
| Critical Ratio :   0.763        | Criteria: Flexure                          |
| Critical Clause:   9.5.2                                                   |

---

CROSS SECTION

| Shape: Tee-Shape | YD: 2.17 | ZD: 2.50 | ZB: 0.83 | YB: 1.83 |

---

DESIGN INPUTS

| Concrete | Fc: 576.000 |                     | Ec: 0.519E+06 |
| Steel    | Fy(main): 8639.999 | Fy(trans): 8639.999 | Es: 0.418E+07 |
| Cover    | Top: 0.125 | Bottom: 0.125 | Sides: 0.125 |

---

CRITICAL STRENGTH RESULTS

<table>
<thead>
<tr>
<th>Category</th>
<th>Demand</th>
<th>Min Capacity</th>
<th>Max Capacity</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>0.000</td>
<td>-825.395</td>
<td>411.480</td>
<td>0.000</td>
</tr>
<tr>
<td>Flexure</td>
<td>533.333</td>
<td>-23.984</td>
<td>698.938</td>
<td>0.763</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>-17.485</td>
<td>17.485</td>
<td>0.000</td>
</tr>
<tr>
<td>Shear Z</td>
<td>0.000</td>
<td>-8.467</td>
<td>8.467</td>
<td>0.000</td>
</tr>
<tr>
<td>Torsion</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

---

LONGITUDINAL BAR DETAILS AT CROSS SECTIONS

---

<table>
<thead>
<tr>
<th>Distance</th>
<th>Position</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
</table>
---
V. ACI 318-14 Rectangular Beam without Torsion

Design of a doubly reinforced rectangular beam.

Reference
1. ACI 318-14
2. PCA Notes 2011 Example 6.2

Details
To determine the Required Area of Tension and Compression Reinforcement for given Concrete and Steel Properties with Given Factored Moment.

Design of Rectangular Beam with Compression Reinforcement-Doubly Reinforced

A beam cross-section is limited to size 14in X 23 in. Determine the required area of reinforcement for a factored moment Mu= 516 ft-kips. fc= 4000psi, fy=60,000 psi.

Design Properties, available:

Validation
\( d_{cover} = 2.5 \text{ in} \)

Step 1: Check if compression reinforcement is required, using \( Z = 0.9 \).
See table 6-1 of PCA Notes (ref. 2).

\[ b = 14 \text{ in} \]
\[ d = 20.5 \text{ in} \]

**Note:** Assume \( d = d_t \)

\[ M_n = M_u/Z = 573.33 \text{ ft-kip} \]
\[ R_n = M_n / (b^2d^2) = 1.169(10)^3 \text{ psi} \]
\[ R_{nt} = 911 \text{ psi} \]

**Note:** \( R_{nt} = \text{Maximum } R_n \text{ for tension controlled section for that particular grade for concrete.} \)

\[ R_n > R_{nt} \]

Hence, both compression and tension reinforcement is required.

**Step 2:** Determine maximum moment with compression reinforcement,

\[ M_{nt} \]

This condition corresponds to tension controlled limit (\( Z = 0.9 \)).

Depth of neutral axis, \( c = 0.375d_t = 0.375d = 7.687 \text{ in} \)
\[ \beta_1 = 0.85 \text{ (from table 6.1 (ref. 2))} \]
\( a = \beta_1c = 6.534 \text{ in} \)

Compressive force, \( C = 0.85 f_c \cdot a \cdot b = 311.036 \text{ kips} \)

Tension force, \( T = C = 311.036 \text{ kips} \)

Nominal moment strength resisted by the concrete section, without compression reinforcement, \( M_{nt} = T(d - a/2) = 446.669 \text{ ft-kips} \)

Nominal moment strength resisted by the concrete section, with compression reinforcement, \( M_{nc} = M_n \cdot M_{nt} = 126.664 \text{ ft-kips} \)

**Step 3: Determine area of tension steel**

Area of tension steel required, \( A_{s,nt} \), to develop \( M_{nt} \)

\[ A_{s,nt} = \frac{T}{f_y} = 5.184 \text{ in}^2 \]

**Step 4: Determine additional area of tension steel required**

Area of tension steel required, \( A_{s,\text{additional}} \), required to counteract additional moment due to T-C couple.

\[ A_{s,\text{additional}} = \frac{(M_n \cdot M_{nt})}{[f_y \cdot (d - d_{\text{cover}})]} = 1.407 \text{ in}^2 \]

Total tension steel required:

\[ A_s = A_s + A_{s,nt} = 6.591 \text{ in}^2 \]

**Comparison**

Table 642: Comparison of results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>STAAD.Pro</th>
<th>Reference</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area Required (compression) (in²)</td>
<td>1.441</td>
<td>1.61</td>
<td>11.0</td>
<td>See note (*) below.</td>
</tr>
<tr>
<td>Area Required (tension) (in²)</td>
<td>6.591</td>
<td>6.355</td>
<td>3.6%</td>
<td>See note (*) below.</td>
</tr>
</tbody>
</table>

**Note:** (*) The required steel calculation above assumes a single layer of bottom bars and the effective depth = 0.375*d*t based on the single bar layer. The reference (ref. 2) calculations only considers a strength design and does not check for the minimum spacing requirements. STAAD.Pro however considers two layers of bars in the design to account for the minimum criteria and hence uses a neutral axis position based on the two bar layers. Hence the minor difference in the steel areas calculated.
JOB NAME PCA Example 6.2
JOB CLIENT Bentley Systems Inc.
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 8 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 1.91667 ZD 1.16667
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
JOINT LOAD
2 MZ 464.417
PERFORM ANALYSIS
PRINT SUPPORT REACTION ALL
START CONCRETE DESIGN
CODE ACI
UNIT INCHES KIP
MAXMAIN 8 ALL
MINMAIN 8 ALL
*MIMB 6 ALL
*MXMB 6 ALL
CLT 2 ALL
CLB 2 ALL
TRACK 2 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH

**STAAD Output**

<table>
<thead>
<tr>
<th>Status</th>
<th>Pass</th>
<th>Type</th>
<th>Beam</th>
<th>Length: 96.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical Ratio</td>
<td>0.910</td>
<td>Criteria: Flexure</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Critical Clause</td>
<td>9.5.2</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
CROSS SECTION

| Shape: Rectangular | Width: 14.00 | Depth: 23.00 |

DESIGN INPUTS

| Concrete | Fc          | 4.000 |                     | Ec     | 0.360E+04 |
| Steel    | Fy(main)    | 60.000 | Fy(trans)    60.000 | Es     | 0.290E+05 |
| Cover    |             | Top    |                      | Bottom | 2.000 |
|          |             | Sides  |                      |        | 1.500 |

CRITICAL STRENGTH RESULTS

<table>
<thead>
<tr>
<th>Category</th>
<th>Demand</th>
<th>Min Capacity</th>
<th>Max Capacity</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>0.000</td>
<td>-871.565</td>
<td>554.580</td>
<td>0.000</td>
</tr>
<tr>
<td>Flexure</td>
<td>5573.004</td>
<td>-2586.136</td>
<td>6122.188</td>
<td>0.910</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>-21.019</td>
<td>22.720</td>
<td>0.000</td>
</tr>
<tr>
<td>Shear Z</td>
<td>0.000</td>
<td>-22.720</td>
<td>22.720</td>
<td>0.000</td>
</tr>
<tr>
<td>Torsion</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

LONGITUDINAL BAR DETAILS AT CROSS SECTIONS

<table>
<thead>
<tr>
<th>Distance</th>
<th>Position</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>Top</td>
<td>1.610</td>
<td>2.370</td>
<td>3</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>6.355</td>
<td>7.900</td>
<td>10</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td>24.000</td>
<td>Top</td>
<td>1.610</td>
<td>2.370</td>
<td>3</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>6.355</td>
<td>7.900</td>
<td>10</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td>48.000</td>
<td>Top</td>
<td>1.610</td>
<td>2.370</td>
<td>3</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>6.355</td>
<td>7.900</td>
<td>10</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td>72.000</td>
<td>Top</td>
<td>1.610</td>
<td>2.370</td>
<td>3</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>6.355</td>
<td>7.900</td>
<td>10</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td>96.000</td>
<td>Top</td>
<td>1.610</td>
<td>2.370</td>
<td>3</td>
<td># 8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>6.355</td>
<td>7.900</td>
<td>10</td>
<td># 8</td>
<td></td>
</tr>
</tbody>
</table>

STAAD SPACE  -- PAGE NO.

LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Bars Nums</th>
<th>Size</th>
<th>Location Start</th>
<th>Distance From Face</th>
<th>Anchor Start End</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>3</td>
<td># 8</td>
<td>0.00</td>
<td>96.00</td>
<td>Yes Yes</td>
</tr>
</tbody>
</table>
V. India

V. IS 13920 2016

V. IS13920 2016-Singly Reinforced Rectangular Beam

Verify the design of a rectangular beam per IS 13920:2016.

Details

A three story, concrete structure is subject to gravity loads only. Beam 59 is to be designed:
Section properties:
- Length of beam, \( L = 3,000 \text{ mm} \) (clear span)
- Width of beam, \( B = 250 \text{ mm} \)
- Depth of beam, \( D = 300 \text{ mm} \)
- Clear cover = 30 mm
- Diameter of main reinforcement = 12 mm
- Diameter of stirrups = 8 mm

Material properties:
- Characteristic strength of concrete, \( f_{ck} = 20 \text{ N/mm}^2 \)
- Yield strength of steel, \( f_y = 415 \text{ N/mm}^2 \)

Loads: total uniform force = 25 kN/m

**Validation**

Effective Depth, \( d = 300 - 30 - 8 - (12/2) = 256 \text{ mm} \)

Depth of Compression Reinforcement, \( d' = 30 + (12/2) + 8 = 44 \text{ mm} \)

Check for Member Size

Width to Depth ratio of the member, \( B/D = 250/300 = 0.833 \) (Clause 6.1.1; IS 13920:2016)
Width of Beam, \( B = 250 \, mm > 200 \, mm \) (Clause 6.1.2; IS 13920:2016)

As per clause 6.1.3 of IS 13920:2016, depth of beam shall not have more than 1/4 of clear span. So, the allowable depth not more than, \( L/4 = 3000/4 = 750 \, mm > 300 \, mm \). Hence ok. (Clause 6.1.3; IS 13920:2016)

Width of the supporting member, \( C2 = 400 \, mm \) and Breadth of the supporting member, \( C1 = 400 \, mm \). So, the width of member, \( B \), shall not exceed smaller of \( (C2) \) and \( (C1) = 400 \, mm \) or \( C2 + \) smaller of \( (C1) \) and \( (C2) = 800 \, mm \). Hence ok. (Clause 6.1.4; IS 13920:2016)

Check for Axial Stress

Axial force from STAAD.Pro analysis, \( F_x = 0.299 \, kN \)

Factored axial compressive stress: \( F_x / (B \times D) = 0.004 < 0.08f_{ck} = 1.6 \), Hence OK (Clause 6.1; IS 13920:2016)

Design for Flexure

Minimum longitudinal reinforcement, \( p_{min} = 0.24 \times \sqrt{f_{ck}} / f_y \times B \times d = 165.5 \, mm^2 \)

Maximum longitudinal reinforcement, \( p_{max} = 0.025 \times B \times d = 1,600 \, mm^2 \)

Limiting moment value, \( M_{z,limit} = 0.138f_{ck} \times B \times d^2 = 45.22 \, kN \cdot m \)

From the STAAD.Pro analysis, \( M_z = 30.28 \, kN \cdot m < M_{z,limit} \) at the start of the member (\( x = 0 \)).

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

- Percentage of tension reinforcement \( p_t = 0.583 \)
- Percentage of compression reinforcement \( p_c = 0.292 \)

Reinforcement at Start of Member (\( x = 0 \)): top bars

\[
A_{st, top, a} = 0.583 \times B \times d / 100 = 373.1. \, mm^2
\]

Provide 4 - 12 mm diameter bars. Actual \( A_{st, top} = 4 \times (\pi/4) \times (12)^2 = 452.4 \, mm^2 \)

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

Reinforcement at Start of Member (\( x = 0 \)): bottom bars

\[
A_{st, bot, a} = 0.292 \times B \times d / 100 = 186.9 \, mm^2
\]

Provide 2 - 12 mm diameter bars. Actual \( A_{st, bot} = 2 \times (\pi/4) \times (12)^2 = 226.2 \, mm^2 \)

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

From the STAAD.Pro analysis, \( M_z = 24.56 \, kN \cdot m < M_{z,limit} \) at the end of the member (\( x = 3 \, m \)).

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

- Percentage of tension reinforcement \( p_t = 0.451 \)
- Percentage of compression reinforcement \( p_c = 0.225 \)

Reinforcement at End of Member (\( x = 3 \, m \)): top bars

\[
A_{st, top, b} = 0.451 \times B \times d / 100 = 288.6 \, mm^2
\]

Provide 3 - 12 mm diameter bars. Actual \( A_{st, top} = 3 \times (\pi/4) \times (12)^2 = 339.3 \, mm^2 \)

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

Reinforcement at End of Member (\( x = 3 \, m \)): bottom bars

\[
A_{st, bot, b} = 0.225 \times B \times d / 100 = 144 \, mm^2
\]
Provide 2 - 12 mm diameter bars. Actual \( A_{st, bot} = 2 \times (\pi/4) \times (12)\times (12) = 226.2 \text{ mm}^2 \)

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

As per clause 6.2.4 of IS 13920:2016, any intermediate section, on top or bottom face should have at least 452.4 \( \text{mm}^2/4 = 113.1 \text{ mm}^2 \) of steel as the main reinforcement. So the provided reinforcement satisfies this condition.

**Design for Shear**

Calculation of resistance moment based on the area of steel provided:

Sagging resistance moment at start:

\[
M_{z,as} = 0.87 \times f_y \times A_{st,bot,a} \times d \times [1 - A_{st,bot,a} \times f_y / (B \times d \times f_{ck})] = 19.37 \text{ kN m}
\]

Hogging resistance moment at start:

\[
M_{z,ah} = 0.87 \times f_y \times A_{st,top,a} \times d \times [1 - A_{st,top,a} \times f_y / (B \times d \times f_{ck})] = 35.68 \text{ kN m}
\]

Sagging resistance moment at end:

\[
M_{z,bs} = 0.87 \times f_y \times A_{st,bot,b} \times d \times [1 - A_{st,bot,b} \times f_y / (B \times d \times f_{ck})] = 19.37 \text{ kN m}
\]

Hogging resistance moment at end:

\[
M_{z,bh} = 0.87 \times f_y \times A_{st,top,b} \times d \times [1 - A_{st,top,b} \times f_y / (B \times d \times f_{ck})] = 27.91 \text{ kN m}
\]

Simple shear from gravity load on the span:

\[
V_a = 1.2 \times 25 \text{ kN/m} \times 3/2 = 45 \text{ kN}
\]

Factor shear force as per linear structural analysis in STAAD.Pro, \( F_y = 58.16 \text{ kN} \)

Calculation of shear force due to formation of plastic hinge at both ends of the beam plus the factored gravity load on the span.

For sway to right:

\[
V_{u,ar} = V_a - 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 22.93 \text{ kN}
\]

\[
V_{u,bv} = V_a + 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 67.01 \text{ kN}
\]

For sway to left:

\[
V_{u,al} = V_a + 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 70.69 \text{ kN}
\]

\[
V_{u,bl} = V_a - 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 19.31 \text{ kN}
\]

Design shear force at start: \( V_{ua} = 70.69 \text{ kN} \)

Design shear force at end: \( V_{ub} = 67.01 \text{ kN} \)

Design shear force, \( V_{ua} = 70.69 \text{ kN} \) (Considering clause 6.3.4(c) of IS13920:2016)

Provide 2 legged 8 mm diameter bar. Thus the area of shear reinforcement:

\[
A_v = 2 \times (\pi/4) \times (8)\times (8) = 100.5 \text{ mm}^2
\]

Using the formula from IS 456:2000, the spacing for vertical stirrups is:

\[
S_v = 0.87 \times f_y \times A_{sv} \times d / V_{ub} = 131.4 \text{ mm}
\]

Per clause 6.3.5 of IS13920:2016, the spacing of the links over a length of \( 2d \) at either end of the beam shall be the least of:

- \( d/4 = 64 \text{ mm} \)
b. $6 \times$ bar diameter of the smallest bars = $6 \times 12 = 72$ mm  
c. $100$ mm  
d. $S_v = 131$ mm

Thus, it should be not greater than $64$ mm. Therefore, provide 2 legged - 8 mm links @ 60 mm center to center at left and right end over a length of $2d = 512$ mm.

Per Clause 6.3.5.2 of IS13920:2016, the spacing of the stirrups in the mid-span shall not exceed $d/2 = 128$ mm. Thus, use a spacing of 125 mm elsewhere.

\textbf{Results}

\textbf{Table 643: Comparison of results}

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required area of steel at top of start section (mm$^2$)</td>
<td>373.1</td>
<td>377.0</td>
<td>1.0%</td>
<td></td>
</tr>
<tr>
<td>Required area of steel at bottom of start section (mm$^2$)</td>
<td>186.9</td>
<td>188.5</td>
<td>0.6%</td>
<td></td>
</tr>
<tr>
<td>Required area of steel at top of end section (mm$^2$)</td>
<td>288.6</td>
<td>296.92</td>
<td>2.9%</td>
<td>negligible difference that give same bar selection</td>
</tr>
<tr>
<td>Required area of steel at bottom of end section (mm$^2$)</td>
<td>144.0</td>
<td>148.46</td>
<td>3.1%</td>
<td>negligible difference that give same bar selection</td>
</tr>
<tr>
<td>Spacing of shear reinforcement at ends (mm)</td>
<td>60</td>
<td>60</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

\textbf{STAAD.Pro Input File}

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS 13920\2016\IS13920 2016-Singly Reinforced Rectangular Beam.STD is typically installed with the program.

\textbf{STAAD SPACE}

START JOB INFORMATION  
ENGINEER DATE 25-Jul-18  
END JOB INFORMATION  
INPUT WIDTH 79  
******************************************************************************
*  
*This Problem is created to verify the design results produced by the program  
*with hand calculated design results for a rectangular (300 mm * 250 mm) singly  
*reinforced beam section as per IS13920:2016  

\textbf{STAAD.Pro}  
4285  
User Manual
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 0 6 0; 4 0 9 0; 5 3 9 0; 6 6 9 0; 7 6 6 0; 8 6 3 0;
9 6 0 0; 10 3 6 0; 11 3 3 0; 12 3 0 0; 13 0 0 3; 14 0 3 3; 15 0 6 3; 16 0 9 3;
17 3 9 3; 18 6 9 3; 19 6 6 3; 20 6 3 3; 21 6 0 3; 22 3 6 3; 23 3 3 3; 24 3 0 3;
25 0 0 6; 26 0 3 6; 27 0 6 6; 28 0 9 6; 29 3 9 6; 30 6 9 6; 31 6 6 6; 32 6 3 6;
33 6 0 6; 34 3 6 6; 35 3 3 6; 36 3 0 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 3 10; 10 10 7;
11 2 11; 12 11 8; 13 5 10; 14 10 11; 15 11 12; 16 2 14; 17 3 15; 20 4 16;
21 5 17; 22 6 18; 23 7 19; 24 8 20; 25 10 22; 26 11 23; 27 13 14; 28 14 15;
29 15 16; 30 16 17; 31 17 18; 32 18 19; 33 19 20; 34 20 21; 35 15 22; 36 22 19;
37 14 23; 38 23 20; 39 17 22; 40 22 23; 41 23 24; 42 14 26; 43 15 27; 44 16 28;
45 17 29; 46 18 30; 47 19 31; 48 20 32; 49 22 34; 50 23 35; 51 25 26; 52 26 27;
53 27 28; 54 28 29; 55 29 30; 56 30 31; 57 31 32; 58 32 33; 59 27 34; 60 34 31;
61 26 35; 62 35 32; 63 29 34; 64 34 35; 65 35 36;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 PRIS YD 0.3 ZD 0.25
1 TO 3 6 TO 8 13 14 17 27 TO 29 32 TO 34 39 TO 41 51 TO 53 56 TO 58 -
63 TO 65 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 9 12 13 21 24 25 33 36 FIXED
LOAD 1 LOADTYPE Dead TITLE DL
MEMBER LOAD
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -3.125
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -1.875
LOAD 2 LOADTYPE Live TITLE LL
MEMBER LOAD
4 5 20 22 44 46 54 55 UNI GY -7.875
21 30 31 45 UNI GY -15.75
9 TO 12 18 19 23 24 42 43 47 48 59 TO 62 UNI GY -20
25 26 35 TO 38 49 50 UNI GY -90
LOAD COMB 3 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 1
1 1.5 2 1.5
LOAD COMB 4 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 2
1 1.2 2 1.2
LOAD COMB 5 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 3
STAAD.Pro Output

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>FLEXURE (Maxm. Sagging/Hogging moments)</th>
<th>SHEAR</th>
<th>VY</th>
<th>MX Load</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>P</td>
<td>MZ</td>
<td>MX</td>
<td>Load Case</td>
</tr>
<tr>
<td>0.0</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td>250.0</td>
<td>0.00</td>
<td>-30.28</td>
<td>0.00</td>
<td>3</td>
</tr>
<tr>
<td>500.0</td>
<td>0.00</td>
<td>-16.91</td>
<td>0.00</td>
<td>3</td>
</tr>
<tr>
<td>750.0</td>
<td>0.00</td>
<td>-5.89</td>
<td>0.00</td>
<td>3</td>
</tr>
<tr>
<td>1000.0</td>
<td>0.00</td>
<td>2.79</td>
<td>0.00</td>
<td>3</td>
</tr>
<tr>
<td>1250.0</td>
<td>0.00</td>
<td>13.12</td>
<td>0.00</td>
<td>3</td>
</tr>
</tbody>
</table>

Verification Examples

STAAD.Pro 4287 User Manual
**Verification Examples**

V.10 Concrete Design

---

**STAAD SPACE**

---

### SUMMARY OF REINF. AREA (Sq.mm)

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>TOP Reqd./Provided reinf.</th>
<th>BOTTOM Reqd./Provided reinf.</th>
<th>STIRRUPS (2 legged)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>377.00/452.39(4-12d)</td>
<td>188.50/226.19(2-12d)</td>
<td>8d @ 60 mm</td>
</tr>
<tr>
<td>250.0</td>
<td>197.26/226.19(2-12d)</td>
<td>94.25/226.19(2-12d)</td>
<td>8d @ 60 mm</td>
</tr>
<tr>
<td>500.0</td>
<td>164.23/226.19(2-12d)</td>
<td>94.25/226.19(2-12d)</td>
<td>8d @ 60 mm</td>
</tr>
<tr>
<td>750.0</td>
<td>94.25/226.19(2-12d)</td>
<td>164.23/226.19(2-12d)</td>
<td>8d @ 120 mm</td>
</tr>
<tr>
<td>1000.0</td>
<td>94.25/226.19(2-12d)</td>
<td>164.23/226.19(2-12d)</td>
<td>8d @ 120 mm</td>
</tr>
<tr>
<td>1250.0</td>
<td>94.25/226.19(2-12d)</td>
<td>164.23/226.19(2-12d)</td>
<td>8d @ 120 mm</td>
</tr>
<tr>
<td>1500.0</td>
<td>94.25/226.19(2-12d)</td>
<td>170.69/226.19(2-12d)</td>
<td>8d @ 120 mm</td>
</tr>
<tr>
<td>1750.0</td>
<td>94.25/226.19(2-12d)</td>
<td>164.23/226.19(2-12d)</td>
<td>8d @ 120 mm</td>
</tr>
<tr>
<td>2000.0</td>
<td>94.25/226.19(2-12d)</td>
<td>164.23/226.19(2-12d)</td>
<td>8d @ 120 mm</td>
</tr>
<tr>
<td>2250.0</td>
<td>94.25/226.19(2-12d)</td>
<td>164.23/226.19(2-12d)</td>
<td>8d @ 120 mm</td>
</tr>
<tr>
<td>2500.0</td>
<td>164.23/226.19(2-12d)</td>
<td>94.25/226.19(2-12d)</td>
<td>8d @ 60 mm</td>
</tr>
<tr>
<td>2750.0</td>
<td>164.23/226.19(3-12d)</td>
<td>148.46/226.19(2-12d)</td>
<td>8d @ 60 mm</td>
</tr>
<tr>
<td>3000.0</td>
<td>296.92/339.29(3-12d)</td>
<td>148.46/226.19(2-12d)</td>
<td>8d @ 60 mm</td>
</tr>
</tbody>
</table>

---

**SHEAR DESIGN RESULTS AT DISTANCE 2d (TWICE EFFECTIVE DEPTH) FROM FACE OF THE SUPPORT**

**SHEAR DESIGN RESULTS AT 708.0 mm AWAY FROM START SUPPORT**

\[ V_Y = 31.76 \text{ kN} \]

Provide 2 Legged 8d @ 60 mm c/c

**SHEAR DESIGN RESULTS AT 708.0 mm AWAY FROM END SUPPORT**

\[ V_Y = -27.94 \text{ kN} \]

Provide 2 Legged 8d @ 60 mm c/c
V.IS13920 2016-Doubly Reinforced Rectangular Beam

Verify the design of a rectangular beam per IS 13920:2016.

Details

A three story, concrete structure is subject to gravity loads only. Beam 59 is to be designed:

Length of beam, \( L = 3,000 \, \text{mm} \) (clear span)
Width of beam, \( B = 250 \, \text{mm} \)
Depth of beam, \( D = 300 \, \text{mm} \)
Clear cover = 30 \, \text{mm}
Diameter of main reinforcement = 16 \, \text{mm}
Diameter of stirrups = 8 \, \text{mm}
Material properties:

- Characteristic strength of concrete, $f_{ck} = 20 \text{ N/mm}^2$
- Yield strength of steel, $f_y = 415 \text{ N/mm}^2$

Loads: total uniform force = $50 \text{ kN/m}$

**Validation**

Effective Depth, $d = 300 - 30 - 8 - (16/2) = 254 \text{ mm}$

Depth of Compression Reinforcement, $d' = 30 + (16/2) + 8 = 46 \text{ mm}$

*Check for Member Size*

Width to Depth ratio of the member, $B/D = 250/300 = 0.833$ (Clause 6.1.1; IS 13920:2016)

Width of Beam, $B = 250 \text{ mm} > 200 \text{ mm}$ (Clause 6.1.2; IS 13920:2016)

As per clause 6.1.3 of IS 13920:2016, depth of beam shall not have more than $1/4$ of clear span. So, the allowable depth not more than, $L/4 = 3000/4 = 750 \text{ mm} > 300 \text{ mm}$. Hence ok. (Clause 6.1.3; IS 13920:2016)

Width of the supporting member, $C_2 = 400 \text{ mm}$ and Breadth of the supporting member, $C_1 = 400 \text{ mm}$. So, the width of member, $B$, shall not exceed smaller of $(C_2)$ and $(C_1) = 400 \text{ mm} or (C_2 + smaller of (C_1) and (C_2) = 800 \text{ mm}$. Hence ok. (Clause 6.1.4; IS 13920:2016)

Axial force from STAAD.Pro analysis, $F_x = 6.805 \text{ kN}$

Factored axial compressive stress: $F_x / (B \times D) = 0.09 < 0.08 f_{ck} = 1.6$, Hence OK (Clause 6.1; IS 13920:2016)

**Design for Flexure**

Minimum longitudinal reinforcement, $p_{min} = 0.24 \times (f_{ck} / f_y) \times B \times d = 164.2 \text{ mm}^2$

Maximum longitudinal reinforcement, $p_{max} = 0.025 \times B \times d = 1,588 \text{ mm}^2$

Limiting moment value, $M_{z,limit} = 0.138 f_{ck} \times B \times d^2 = 44.52 \text{ kN} \cdot \text{m}$

From the STAAD.Pro analysis, $M_z = 55.44 \text{ kN} \cdot \text{m} > M_{z,limit}$ at the start of the member ($x = 0$).

So, the actual moment is greater than limiting moment then the section will be design as doubly reinforced section.

From SP16 table 50 flexure, reinforcement percentage for doubly reinforced section:

- Percentage of tension reinforcement $p_t = 1.19$
- Percentage of compression reinforcement $p_c = 0.595$

**Reinforcement at Start of Member ($x = 0$): top bars**

$$A_{st, top.a} = 1.19 \times B \times d / 100 = 755.7 \text{ mm}^2$$

Provide 4 - 16 mm diameter bars. Actual $A_{st, top} = 4 \times (\pi/4) \times (16)^2 = 804.2 \text{ mm}^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

**Reinforcement at Start of Member ($x = 0$): bottom bars**

$$A_{st, bot.a} = 0.595 \times B \times d / 100 = 377.8 \text{ mm}^2$$

Provide 2 - 16 mm diameter bars. Actual $A_{st, bot} = 2 \times (\pi/4) \times (16)^2 = 402.1 \text{ mm}^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.
From the STAAD.Pro analysis, \( M_z = 54.11 \text{ kN·m} > M_{z,\text{limit}} \) at the end of the member (\( x = 3 \text{ m} \)).

From SP16 table 50 flexure, reinforcement percentage for doubly reinforced section

Percentage of tension reinforcement \( p_t = 1.161 \) and compression reinforcement \( p_c = 0.581 \).

Reinforcement at End of Member (\( x = 3 \text{ m} \)):

- **Top bars**
  \[ A_{st,\text{top, b}} = 1.161 \times B \times d / 100 = 737.4 \text{ mm}^2 \]
  
  Provide 4 - 16 mm diameter bars. Actual \( A_{st, \text{top}} = 3 \times (\pi/4) \times (16)^2 = 804.2 \text{ mm}^2 \)

  Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

- **Bottom bars**
  \[ A_{st,\text{bot, b}} = 0.225 \times B \times d / 100 = 368.7 \text{ mm}^2 \]
  
  Provide 2 - 16 mm diameter bars. Actual \( A_{st, \text{bot}} = 2 \times (\pi/4) \times (16)^2 = 402.1 \text{ mm}^2 \)

  Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

As per clause 6.2.4 of IS 13920:2016, any intermediate section, on top or bottom face should have at least \( 804.2 \text{ mm}^2 / 4 = 201.1 \text{ mm}^2 \) of steel as the main reinforcement. So the provided reinforcement satisfies this condition.

**Design for Shear**

Calculation of resistance moment based on the area of steel provided:

Sagging resistance moment at start:

\[ M_{z,\text{as}} = 0.87 \times f_y \times A_{st,\text{bot, a}} \times d \times \left[ 1 - A_{st,\text{bot, a}} \times f_y / (B \times d \times f_{ck}) \right] = 32.03 \text{ kN·m} \]

Hogging resistance moment at start:

\[ M_{z,\text{ah}} = 0.87 \times f_y \times A_{st,\text{top, a}} \times d \times \left[ 1 - A_{st,\text{top, a}} \times f_y / (B \times d \times f_{ck}) \right] = 54.37 \text{ kN·m} \]

Sagging resistance moment at end:

\[ M_{z,\text{bs}} = 0.87 \times f_y \times A_{st,\text{bot, b}} \times d \times \left[ 1 - A_{st,\text{bot, b}} \times f_y / (B \times d \times f_{ck}) \right] = 32.03 \text{ kN·m} \]

Hogging resistance moment at end:

\[ M_{z,\text{bh}} = 0.87 \times f_y \times A_{st,\text{top, b}} \times d \times \left[ 1 - A_{st,\text{top, b}} \times f_y / (B \times d \times f_{ck}) \right] = 54.37 \text{ kN·m} \]

Simple shear from gravity load on the span:

\[ V_a = 1.2 \times 50 \text{ kN/m} \times 3/2 = 90 \text{ kN} \]

Factor shear force as per linear structural analysis in STAAD.Pro, \( F_y = 112.9 \text{ kN} \)

Calculation of shear force due to formation of plastic hinge at both ends of the beam plus the factored gravity load on the span.

For sway to right:

\[ V_{u,\text{ar}} = V_a - 1.4 \times (M_{z,\text{as}} + M_z \times b \times h/3) = 49.68 \text{ kN} \]

\[ V_{u,\text{br}} = V_a + 1.4 \times (M_{z,\text{as}} + M_z \times b \times h/3) = 130.3 \text{ kN} \]

For sway to left:

\[ V_{u,\text{al}} = V_a + 1.4 \times (M_{z,\text{as}} + M_z \times b \times h/3) = 130.3 \text{ kN} \]

\[ V_{u,\text{bl}} = V_a - 1.4 \times (M_{z,\text{as}} + M_z \times b \times h/3) = 49.68 \text{ kN} \]

Design shear force at start: \( V_{ua} = 130.3 \text{ kN} \)
Design shear force at end: \( V_{ub} = 130.3 \text{ kN} \)

Design shear force, \( V_{ua} = 130.3 \text{ kN} \) (Considering clause 6.3.4(c) of IS13920:2016)

Provide 2 legged 8 mm diameter bar. Thus the area of shear reinforcement:

\[
A_v = 2 \times \left( \frac{\pi}{4} \right) \times 8^2 = 100.5 \text{ mm}^2
\]

Using the formula from IS 456:2000, the spacing for vertical stirrups is:

\[
S_v = 0.87 \times f_y \times A_{sv} \times d / V_{ua} = 70.7 \text{ mm}
\]

Per clause 6.3.5 of IS13920:2016, the spacing of the links over a length of \( 2d \) at either end of the beam shall be the least of:

a. \( d/4 = 64 \text{ mm} \)

b. \( 6 \times \text{bar diameter of the smallest bars} = 6 \times 16 = 96 \text{ mm} \)

c. \( 100 \text{ mm} \)

d. \( S_v = 71 \text{ mm} \)

Thus, it should be not greater than 64 mm. Therefore, provide 2 legged - 8 mm links @ 60 mm center to center at left and right end over a length of \( 2d = 508 \text{ mm} \).

Per Clause 6.3.5.2 of IS13920:2016, the spacing of the stirrups in the mid-span shall not exceed \( d/2 = 127 \text{ mm} \). Thus, use a spacing of 125 mm elsewhere.

Results

Table 644: Comparison of results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required area of steel at top of start section (mm(^2))</td>
<td>769.4</td>
<td>772.79</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Required area of steel at bottom of start section (mm(^2))</td>
<td>384.7</td>
<td>386.39</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Required area of steel at top of end section (mm(^2))</td>
<td>751.2</td>
<td>754.73</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Required area of steel at bottom of end section (mm(^2))</td>
<td>375.6</td>
<td>377.37</td>
<td>negligible</td>
<td></td>
</tr>
<tr>
<td>Spacing of shear reinforcement at ends (mm)</td>
<td>60</td>
<td>60</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>
Verification Examples
V.10 Concrete Design

STAAD.Pro Input File

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS 13920\2016\IS13920 2016-Doubly Reinforced Rectangular Beam.STD is typically installed with the program.

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Jul-18
END JOB INFORMATION
INPUT WIDTH 79
******************************************************************************
* This Problem is created to verify the design results produced by the program
* with hand calculated design results for a rectangular(300 mm * 250 mm)
doubly*
*reinforced beam section as per IS13920:2016
*
*
******************************************************************************
* UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 0 6 0; 4 0 9 0; 5 3 9 0; 6 6 9 0; 7 6 6 0; 8 6 3 0;
9 6 0 0; 10 3 6 0; 11 3 3 0; 12 3 0 0; 13 0 0 3; 14 0 3 3; 15 0 6 3; 16 0 9 3;
17 3 9 3; 18 6 9 3; 19 6 6 3; 20 6 3 3; 21 6 0 3; 22 3 6 3; 23 3 3 3; 24 3 0
3;
25 0 0 6; 26 0 3 6; 27 0 6 6; 28 0 9 6; 29 3 9 6; 30 6 9 6; 31 6 6 6; 32 6 3
6;
33 6 0 6; 34 3 6 6; 35 3 3 6; 36 3 0 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 3 10; 10 10 7;
11 2 11; 12 11 8; 13 5 10; 14 10 11; 17 11 12; 18 2 14; 19 3 15; 20 4 16;
21 5 17; 22 6 18; 23 7 19; 24 8 20; 25 10 22; 26 11 23; 27 13 14; 28 14 15;
29 15 16; 30 16 17; 31 17 18; 32 18 19; 33 19 20; 34 20 21; 35 21 22; 36 22
19;
37 14 23; 38 23 20; 39 17 22; 40 22 23; 41 23 24; 42 14 26; 43 15 27; 44 16
28;
45 17 29; 46 18 30; 47 19 31; 48 20 32; 49 22 34; 50 23 35; 51 25 26; 52 26
27;
53 27 28; 54 28 29; 55 29 30; 56 30 31; 57 31 32; 58 32 33; 59 27 34; 60 34
31;
61 26 35; 62 35 32; 63 29 34; 64 34 35; 65 35 36;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 PRIS YD 0.3 ZD
0.25
### Verification Examples

#### V.10 Concrete Design

---

<table>
<thead>
<tr>
<th>1 TO 3 6 TO 8 13 14 17 27 TO 29 32 TO 34 39 TO 41 51 TO 53 56 TO 58 - 63 TO 65 PRIS YD 0.4 ZD 0.4</th>
</tr>
</thead>
</table>

**CONSTANTS**

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>CONCRETE ALL</th>
</tr>
</thead>
</table>

**SUPPORTS**

1 9 12 13 21 24 25 33 36 FIXED

**LOAD 1**

<table>
<thead>
<tr>
<th>LOADTYPE</th>
<th>Dead</th>
<th>TITLE</th>
<th>DL</th>
</tr>
</thead>
</table>

**MEMBER LOAD**

| 4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY | -3.125 |
| 4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY | -1.875 |

**LOAD 2**

<table>
<thead>
<tr>
<th>LOADTYPE</th>
<th>Live</th>
<th>TITLE</th>
<th>LL</th>
</tr>
</thead>
</table>

**MEMBER LOAD**

| 4 5 20 22 44 46 54 55 UNI GY | -7.875 |
| 21 30 31 45 UNI GY | -15.75 |
| 9 TO 12 18 19 23 24 42 43 47 48 59 TO 62 UNI GY | -45 |
| 25 26 35 TO 38 49 50 UNI GY | -90 |

**LOAD COMB 3**

| GENERATED IS:456/IS:800 GENRAL_STRUCTURES | 1 1.5 2 1.5 |

**LOAD COMB 4**

| GENERATED IS:456/IS:800 GENRAL_STRUCTURES | 1 1.2 2 1.2 |

**LOAD COMB 5**

| GENERATED IS:456/IS:800 GENRAL_STRUCTURES | 1 1.5 |

**LOAD COMB 6**

| GENERATED IS:456/IS:800 GENRAL_STRUCTURES | 1 0.9 |

**LOAD COMB 7**

| COMBINATION LOAD CASE 7 | 1 1.0 2 1.0 |

**PERFORM ANALYSIS**

**PRINT**

**SUPPORT REACTION**

**PRINT MEMBER FORCES**

**START CONCRETE DESIGN**

**CODE IS13920 2016**

| GLD | 7 MEMB 59 |
|-------------------|
| CLB | 0.03 ALL |
| CLT | 0.03 ALL |

| FC | 20000 MEMB 59 |
| FYNAM | 415000 MEMB 59 |
| FYS | 415000 MEMB 59 |

**MAXMAIN**

| 16 MEMB 59 |
| MAXSEC | 8 MEMB 59 |

**MINMAIN**

| 16 MEMB 59 |
| MINSEC | 8 MEMB 59 |

**TRACK**

| 2 MEMB 59 |

**DESIGN BEAM 59**

**END CONCRETE DESIGN**

**FINISH**

---

### STAAD.Pro Output

<table>
<thead>
<tr>
<th>IS - 13920 BEAM DESIGN RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>EUDL CONSIDERED ON MEMBER # 59 IS 50.00 N/MM.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>IS-13920 LIMIT STATE DESIGN BEAM NO. 59 DESIGN RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>M20 Fe415 (Main) Fe415 (Sec.)</td>
</tr>
</tbody>
</table>

---
### Design Load Summary (KN Met)

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>FLEXURE (Maxm. Sagging/Hogging moments)</th>
<th>SHEAR Load</th>
<th>Case</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>P</td>
<td>MZ</td>
<td>MX</td>
</tr>
<tr>
<td>0.0</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>250.0</td>
<td>0.00</td>
<td>-55.44</td>
<td>0.77</td>
</tr>
<tr>
<td>500.0</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>750.0</td>
<td>0.00</td>
<td>8.17</td>
<td>0.77</td>
</tr>
<tr>
<td>1000.0</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1250.0</td>
<td>0.00</td>
<td>27.14</td>
<td>0.77</td>
</tr>
<tr>
<td>1500.0</td>
<td>0.00</td>
<td>29.60</td>
<td>0.77</td>
</tr>
<tr>
<td>1750.0</td>
<td>0.00</td>
<td>27.36</td>
<td>0.77</td>
</tr>
<tr>
<td>2000.0</td>
<td>0.00</td>
<td>20.44</td>
<td>0.77</td>
</tr>
<tr>
<td>2250.0</td>
<td>0.00</td>
<td>8.84</td>
<td>0.77</td>
</tr>
<tr>
<td>2500.0</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2750.0</td>
<td>0.00</td>
<td>-7.46</td>
<td>0.77</td>
</tr>
<tr>
<td>3000.0</td>
<td>0.00</td>
<td>-54.11</td>
<td>0.77</td>
</tr>
</tbody>
</table>

*** Design Shear Force at Section 0.0 is 133.30 KN.***

- Clause 6.3.3 of IS-13920

*** Design Shear Force at Section 3000.0 is 133.30 KN.***

- Clause 6.3.3 of IS-13920

Note:
Moment of resistance is calculated based on the area of steel provided. If area of steel provided is much higher compared to area of steel required moment of resistance will increase which may increase design shear force.

### Summary of Reinforcement Area (Sq.mm)

<table>
<thead>
<tr>
<th>SECTION (in mm)</th>
<th>TOP Req'd/Provided reinf.</th>
<th>BOTTOM Req'd/Provided reinf.</th>
<th>STIRRUPS (2 legged)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>772.79/ 804.25( 4-16d)</td>
<td>386.39/ 402.12( 2-16d)</td>
<td>8d @ 60 mm</td>
</tr>
<tr>
<td>250.0</td>
<td>384.93/ 402.12( 2-16d)</td>
<td>193.20/ 402.12( 2-16d)</td>
<td>8d @ 60 mm</td>
</tr>
<tr>
<td>500.0</td>
<td>193.20/ 402.12( 2-16d)</td>
<td>193.20/ 402.12( 2-16d)</td>
<td>8d @ 60 mm</td>
</tr>
</tbody>
</table>
V.11 Timber Design

V. Canada

V. Timber Design per CAN/CSA 086-01

These verification examples are included for reference purposes.

V. CSA 086 2001 - Glulam in Compression

Reference


Given

Length = 9,000 mm
### Comparison

#### Table 645: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Strength (kN)</td>
<td>295</td>
<td>293.793</td>
<td>none</td>
</tr>
</tbody>
</table>

#### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086\CSA 086 2001 - Glulam in Compression.STD is typically installed with the program.

```plaintext
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 10-JUN-05
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 9 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT_SPRUCE-PINE-12C-E
E 9.7
POISSON 0.15
DENSITY 1.44676e-005
ALPHA 5.5e-006
END DEFINE MATERIAL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 PRIS YD 0.748031 ZD 0.574147
UNIT INCHES KIP
CONSTANTS
MATERIAL GLT_SPRUCE-PINE-12C-E MEMB 1
SUPPORTS
1 PINNED
UNIT METER KN
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
2 FY -214
PERFORM ANALYSIS
PARAMETER
CODE TIMBER CANADIAN
KY 0.5 ALL
KZ 0.5 ALL
CHECK CODE ALL
FINISH
```
### STAAD Output

**STAAD.Pro CODE CHECKING - (S086)**

*ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)*

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX          MY             MZ       LOCATION</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>175.00X228.00</td>
<td>CANADIAN GLULAM GRADE:GLT_SPRUCE-PINE-12C-</td>
</tr>
</tbody>
</table>

|                      | 214.00 C | 0.00 | 0.00 | 0.0000 |

| LEZ = | 4500.000 | LEY = | 4500.000 | LUZ = | 9000.000 | LUY = | 9000.000 mm |

KD = 1.000  KH = 1.000  KT = 1.000  KSB = 1.000  KSV = 1.000
KSC = 1.000  K_SCP = 1.000  KSE = 1.000  KST = 1.000  KZB = 1.000
KZV = 1.000  KZT = 1.000  KZCP = 1.000  K_ZC = 1.000  CHIX = 1.000
CV = 1.000  KN = 1.000

**ACTUAL LOADS : (KN-m)**

Pu = 214.000
Tu = 0.000
Muy = 0.000
Muz = 0.000
V = 0.000
SLENDERNESS_Y = 19.737
SLENDERNESS_Z = 25.714

**ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)**

PY = 413.943
PZ = 293.793
T = 0.000
MY = 0.000
MZ = 0.000
V = 0.000
SLENDERNESS = 50.000

### V. CSA 086 2001 - Glulam in Bending

**Reference**

Example 2, page 59, Canadian Wood Design Manual, 2001

**Given**

Length = 7,500 mm, Beam Spacing = 5,000 mm, Standard load condition, Dry service condition, Untreated
Comparison

Table 646: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Strength in Bending (kN-m)</td>
<td>208</td>
<td>208.323</td>
<td>none</td>
</tr>
<tr>
<td>Design Strength in Shear (kN)</td>
<td>101</td>
<td>100.776</td>
<td>none</td>
</tr>
</tbody>
</table>

STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086\CSA 086 2001 - Glulam in Bending.STD is typically installed with the program.

`STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 10-JUN-05
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0
MEMBER INCIDENCES
1 1 2
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT_SPRUCE-PINE-12C-E
E 9.7
POISSON 0.15
DENSITY 1.44676E-005
ALPHA 5.5E-006
ISOTROPIC GLT_D.FIR-L-20F-E
E 12.4
POISSON 0.15
DENSITY 1.44676E-005
ALPHA 5.5E-006
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68E-005
ALPHA 5.5E-006
DAMP 0.05
END DEFINE MATERIAL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 PRIS YD 2.11942 ZD 0.426508
UNIT INCHES KIP
CONSTANTS
MATERIAL GLT_D.FIR-L-20F-E MEMB 1
SUPPORTS
1 2 PINNED
UNIT METER KN`
### Verification Examples

#### V.11 Timber Design

LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -27.1
PERFORM ANALYSIS
PARAMETER
CODE TIMBER CANADIAN
CHECK CODE ALL
FINISH

### STAAD Output

**STAAD.Pro CODE CHECKING - (S086)**

*ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)*

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>1</td>
<td>130.00X646.00 CANADIAN GLULAM GRADE:GLT_D.FIR-L-20F-E</td>
<td>FAIL</td>
<td>CL.5.5.5/6.5.</td>
<td>1.008</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00 T</td>
<td>0.00</td>
<td>0.00</td>
<td>7.5000</td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>LEZ = 7500.000</th>
<th>LEY = 7500.000</th>
<th>LUZ = 7500.000</th>
<th>LUY = 7500.000 mm</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>KD = 1.000</th>
<th>KH = 1.000</th>
<th>KT = 1.000</th>
<th>KSB = 1.000</th>
<th>KSV = 1.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>KSC = 1.000</td>
<td>K_SC = 1.000</td>
<td>KSE = 1.000</td>
<td>KST = 1.000</td>
<td>KZB = 1.000</td>
</tr>
<tr>
<td>KZV = 1.000</td>
<td>KZT = 1.000</td>
<td>KZCP = 1.000</td>
<td>K_ZC = 1.000</td>
<td>CHIX = 1.000</td>
</tr>
<tr>
<td>CV = 1.000</td>
<td>KN = 1.000</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**ACTUAL LOADS : (KN-m)**

<table>
<thead>
<tr>
<th>Pu</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tu</td>
<td>0.000</td>
</tr>
<tr>
<td>Muy</td>
<td>0.000</td>
</tr>
<tr>
<td>Muz</td>
<td>0.000</td>
</tr>
<tr>
<td>V</td>
<td>-101.625</td>
</tr>
</tbody>
</table>

**SLENDERNESS_Y** = 16.932

**SLENDERNESS_Z** = 1.529

**ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)**

<table>
<thead>
<tr>
<th>PY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>PZ</td>
<td>0.000</td>
</tr>
<tr>
<td>T</td>
<td>0.000</td>
</tr>
<tr>
<td>MY</td>
<td>41.923</td>
</tr>
<tr>
<td>MZ</td>
<td>208.323</td>
</tr>
<tr>
<td>V</td>
<td>100.776</td>
</tr>
</tbody>
</table>

**SLENDERNESS** = 50.000

### V. CSA 086 2001 - Glulam in Tension

**Reference**

**Given**

Dry service condition, Untreated

**Comparison**

**Table 647: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Strength in Tension (kN)</td>
<td>257</td>
<td>256.636</td>
<td>none</td>
</tr>
</tbody>
</table>

**STAAD Input**

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086\CSA 086 2001 - Glulam in Tension.STD is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 10-JUN-05
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 9 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT_SPRUCE-PINE-14T-E
E 10.7
POISSON 0.15
DENSITY 1.44676e-05
ALPHA 5.5e-06
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-05
ALPHA 5.5e-06
DAMP 0.05
END DEFINE MATERIAL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 PRIS YD 0.872702 ZD 0.262467
UNIT INCHES KIP
CONSTANTS
MATERIAL GLT_SPRUCE-PINE-14T-E ALL
SUPPORTS
1 PINNED
UNIT METER KN
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
JOINT LOAD
2 FY 250
PERFORM ANALYSIS PRINT STATICS CHECK
```
### V. CSA 086 2001 - Beam in Compression

**Reference**

Example 2, page 113, Canadian Wood Design Manual, 2001

**Given**

Unbraced Length = 5,000 mm
### Comparison

**Table 648: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Strength (kN)</td>
<td>130</td>
<td>129.223</td>
<td>none</td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086\CSA 086 2001 - Beam in Compression.STD is typically installed with the program.

```plaintext
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 08-JUN-05
END JOB INFORMATION
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 0 16.4042 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC DFL_NO2_8X8_POST
E 1.368e+06
POISSON 0.15
DENSITY 25
ALPHA 5.5e-06
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL DFL_NO2_8X8_POST ALL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 TABLE ST DFL_NO2_8X8_POST
SUPPORTS
1 PINNED
2 FIXED BUT FY
UNIT METER KN
LOAD 1 DEAD+LIVE LOAD
JOINT LOAD
2 FY -114
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER 1
CODE TIMBER CANADIAN
KSC 0.91 ALL
K_ZC 1.05 ALL
CHECK CODE ALL
FINISH
```

### STAAD Output

STAAD.Pro CODE CHECKING - (S086)
**************************************************************************
V. CSA 086 2001 - Beam in Bending

Reference
Example 1, page 58, Canadian Wood Design Manual, 2001

Given
Length = 6,000mm, Beam Spacing = 3,000mm, Standard load condition, Dry service condition, Untreated

Comparison
Table 649: Comparison of results

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Strength in Bending (kN-m)</td>
<td>79.8</td>
<td>79.732</td>
<td>none</td>
</tr>
</tbody>
</table>
### Criteria

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Strength in Shear (kN)</td>
<td>46.1</td>
<td>46.170</td>
<td>none</td>
</tr>
</tbody>
</table>

### STAAD Input

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086\CSA 086 2001 - Beam in Bending.STD is typically installed with the program.

**STAAD PLANE**

START JOB INFORMATION

ENGINEER DATE 08-JUN-05

END JOB INFORMATION

UNIT METER KN

JOINT COORDINATES

1 0 0 0; 2 6 0 0; 3 3 0 0;

MEMBER INCIDENCES

1 1 3; 2 3 2;

UNIT FEET POUND

DEFINE MATERIAL START

ISOTROPIC DFL_NO1_10X16_BM

E 1.728e+06

POISSON 0.15

DENSITY 25

ALPHA 5.5e-06

END DEFINE MATERIAL

UNIT METER KN

CONSTANTS

MATERIAL DFL_NO1_10X16_BM ALL

UNIT FEET POUND

MEMBER PROPERTY TIMBER CANADIAN

1 2 TABLE ST DFL_NO1_10X16_BM

SUPPORTS

1 2 FIXED

UNIT METER KN

LOAD 1 DEAD+LIVE LOAD

MEMBER LOAD

1 2 UNI GY -16.4

PERFORM ANALYSIS

PARAMETER 1

CODE TIMBER CANADIAN

KD 1 ALL

K_T 1 ALL

KSB 1 ALL

KZB 0.9 ALL

KZV 0.9 ALL

K_ZC 1.05 ALL

CHECK CODE ALL

FINISH

### STAAD Output

**STAAD.Pro CODE CHECKING - (S086)**

*******************************

---
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST DFL_NO1_10X16_BM</td>
<td>FAIL</td>
<td>CL.5.5.5/6.5.6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00 T</td>
</tr>
<tr>
<td>LEZ = 3000.000 LEY = 3000.000 LUZ = 3000.000 LUY = 3000.000mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>KD = 1.000</td>
<td>KH = 1.000</td>
<td>KT = 1.000</td>
</tr>
<tr>
<td>KSC = 1.000</td>
<td>K_SCP = 1.000</td>
<td>KSE = 1.000</td>
</tr>
<tr>
<td>KZV = 0.900</td>
<td>KZT = 1.000</td>
<td>KZCP = 1.000</td>
</tr>
<tr>
<td>CV = 1.000</td>
<td>KN = 1.000</td>
<td></td>
</tr>
<tr>
<td>ACTUAL LOADS : (KN-m)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pu = 0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tu = 0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Muy = 0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Muz = 49.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>V = 49.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SLENDERNESS_Y = 4.511</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SLENDERNESS_Z = 2.158</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)
| PY = 0.000 |
| PZ = 0.000 |
| T = 0.000 |
| MY = 79.800 |
| MZ = 79.732 |
| V = 46.170 |
| SLENDERNESS = 50.000 |

| 2 ST DFL_NO1_10X16_BM | FAIL | CL.5.5.5/6.5.6 | 1.066 | 1 |
| 0.00 T | 79.200 | 49.20 | 3.000 |
| LEZ = 3000.000 LEY = 3000.000 LUZ = 3000.000 LUY = 3000.000mm |
| KD = 1.000 | KH = 1.000 | KT = 1.000 | KSB = 1.000 | KSV = 1.000 |
| KSC = 1.000 | K_SCP = 1.000 | KSE = 1.000 | KST = 1.000 | KZB = 0.900 |
| KZV = 0.900 | KZT = 1.000 | KZCP = 1.000 | K_ZC = 1.050 | CHIX = 1.000 |
| CV = 1.000 | KN = 1.000 |
| ACTUAL LOADS : (KN-m) |
| Pu = 0.000 |
| Tu = 0.000 |
| Muy = 0.000 |
| Muz = 49.200 |
| V = -49.200 |
| SLENDERNESS_Y = 4.511 |
| SLENDERNESS_Z = 2.158 |

ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)
**V. CSA 086 2001 - Beam in Tension**

*Reference*


*Given*

Dry service condition, Untreated

*Comparison*

**Table 650: Comparison of results**

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Reference</th>
<th>STAAD.Pro</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Strength in Tension (kN)</td>
<td>185</td>
<td>184.338</td>
<td>none</td>
</tr>
</tbody>
</table>

*STAAD Input*

The file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086\CSA 086 2001 - Beam in Tension.STD is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 08-JUN-05
END JOB INFORMATION
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 0 16.4042 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC DFL_NO1_6X8_BM
E 1.728e+06
POISSON 0.15
DENSITY 25
ALPHA 5.5e-06
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL DFL_NO1_6X8_BM ALL
```
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 TABLE ST DFL_NO1_6X8_BM
SUPPORTS
1 PINNED
UNIT METER KN
LOAD 1 DEAD+LIVE LOAD
JOINT LOAD
2 FY 144
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER 1
CODE TIMBER CANADIAN
KH 1.1 ALL
KSC 0.91 ALL
K_ZC 1.05 ALL
CHECK CODE ALL
FINISH

### STAAD Output

STAAD.Pro CODE CHECKING - (S086)

***************

**ALL UNITS ARE - KN METE (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td></td>
</tr>
<tr>
<td>-----------------------------</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1 ST DFL_NO1_6X8_BM</td>
<td>PASS</td>
<td>CL.5.5.10/6.5.12</td>
<td>0.781</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>144.00 T</td>
<td>0.00</td>
<td>0.00</td>
<td>0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>----------</td>
<td>----------------</td>
<td>--------</td>
<td>----------</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LEZ = 5000.000</th>
<th>LEY = 5000.000</th>
<th>LUZ = 5000.000</th>
<th>LUY = 5000.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>KD = 1.000</td>
<td>KH = 1.100</td>
<td>KT = 1.000</td>
<td>KSB = 1.000</td>
</tr>
<tr>
<td>KSC = 0.910</td>
<td>K_SCP = 1.000</td>
<td>KSE = 1.000</td>
<td>KST = 1.000</td>
</tr>
<tr>
<td>KZV = 1.000</td>
<td>KZT = 1.000</td>
<td>KZCP = 1.000</td>
<td>K_ZC = 1.050</td>
</tr>
<tr>
<td>CV = 1.000</td>
<td>KN = 1.000</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ACTUAL LOADS : (KN-m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pu = 0.000</td>
</tr>
<tr>
<td>Tu = -144.000</td>
</tr>
<tr>
<td>Muy = 0.000</td>
</tr>
<tr>
<td>Muz = 0.000</td>
</tr>
<tr>
<td>V = 0.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PY = 0.000</td>
</tr>
<tr>
<td>PZ = 0.000</td>
</tr>
<tr>
<td>T = 184.338</td>
</tr>
<tr>
<td>MY = 0.000</td>
</tr>
<tr>
<td>MZ = 0.000</td>
</tr>
<tr>
<td>V = 0.000</td>
</tr>
</tbody>
</table>
This section contains various examples of analytical models, physical models, macro script files, and more which can be helpful in familiarizing yourself with aspects of STAAD.Pro features.

This UK and US sections include examples of various problems that can be solved using the STAAD engine. These examples represent various structural analyses and design problems commonly encountered by structural engineers.

The example models included here along with many other files are installed with the program by default. The default location for these files is C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\.

**Note:** The option to include these files is selected during the product installation.

### EX. Building Planner Example Models

The following models are included when STAAD.Pro is installed.

These files are located in the C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Building Planner\ folder (default location).

**Table 651: Building Planner examples installed with the program**

<table>
<thead>
<tr>
<th>File name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular Building.std</td>
<td>This model demonstrates how circular shaped non-orthogonal building can be created accurately in the Building Planner workflow. This is a complex model and with the use of plan in dxf format received from architect (included in this folder), the entire plan is created. With the use of tools like Copy, Paste, and Mirror, the symmetrical circular structure can be created by just using one half of the model created in dxf. Columns with different beta angle can be modeled effectively and accurately using the Building Planner.</td>
</tr>
<tr>
<td>Typical Ground +3 Story Building.std</td>
<td>This model is of a typical 4 story building having 3 different plans: Plinth Plan, Typical Plan, and Roof Plan. The Plinth plan consist of beam and column without slabs. The RoofPlan has wall loads only for parapet walls.</td>
</tr>
</tbody>
</table>
Application Examples

EX. CIS/2 Example Models

The following models are included when STAAD.Pro is installed.

These files are located in the
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\CIS2\folder (default location).

Table 652: CIS/2 examples installed with the program

<table>
<thead>
<tr>
<th>File name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>fwp_analysis_model.stp</td>
<td>Analysis model of a three-story steel building with sky light.</td>
</tr>
<tr>
<td>fwp_geom.stp</td>
<td>Geometry-only model of a three-story steel building with sky light.</td>
</tr>
<tr>
<td>sds2_geom.stp</td>
<td>A two by one bay braced frame structure with loads.</td>
</tr>
</tbody>
</table>

Related Links

- *I. To import a CIS/2 file* (on page 2287)

EX. Macro Example Files

The following files are included when STAAD.Pro is installed.

These files are located in the
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Macro\ folder (default location).

Table 653: Macro examples installed with the program

<table>
<thead>
<tr>
<th>File name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>stadiumroof.vbs</td>
<td>A Structure Wizard VBA macro that can be used to parametrically generate a stadium roof model.</td>
</tr>
</tbody>
</table>
tower.vbs
A Structure Wizard VBA macro that can be used to parametrically generate a simple tower model with X braced panels.

Note: These macros are loaded by default when VBA-Macro Models is selected as the Model Type in the Structure Wizard application.

Related Links
• M. VBA Macro Models (on page 726)

EX. OpenSTAAD Example Files
The following files are included when STAAD.Pro is installed.

These files are located in the C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\OpenSTAAD folder (default location).

Table 654: OpenSTAAD examples installed with the program

<table>
<thead>
<tr>
<th>File name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>boxgirder.vbs</td>
<td>An OpenSTAAD macro used to parametrically generate a concrete box girder outline consisting of plate elements.</td>
</tr>
<tr>
<td>concretebeam.vbs</td>
<td>An OpenSTAAD macro used to perform the design of a reinforced concrete beam per ACI 318-99 for a selected member in an analyzed STAAD model.</td>
</tr>
<tr>
<td>Rectangle-Beam.xls</td>
<td>A Microsoft Office Excel spreadsheet which can be used to check the capacity of a rectangular concrete beam with the reinforcement already laid out.</td>
</tr>
<tr>
<td>STAADAndWord.doc</td>
<td>A Microsoft Office Word document that includes a macro to first load the current analytical model from STAAD.Pro and then load the support reaction values for a selected supported node and load case.</td>
</tr>
<tr>
<td>STDandACAD.dwg</td>
<td>A sample AutoCAD drawing file capable of writing the member section forces at any distance along a member in the current analytical model in STAAD.Pro.</td>
</tr>
</tbody>
</table>

Related Links
• OS. To import an existing macro (on page 5052)
• OS. Microsoft Excel Macro (on page 5080)
• OS. Autodesk AutoCAD Macro (on page 5082)
• OS. Microsoft Office Security Options (on page 5085)

EX. Physical Model Examples
The following models are included when STAAD.Pro is installed.
These files are located in the C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Physical Models\ folder (default location).

Table 655: Physical Model examples installed with the program

<table>
<thead>
<tr>
<th>File name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area Loads Example.std</td>
<td>This physical model example showcases how Area loads is applied by creating a panel on the stories. This area load gets assigned to the adjoining members (on the panel’s boundary edges) in terms of member distributed loads. Also, the model contains diaphragm defined for each of the floors. The structure is subjected to load categorized as Dead and Live Load groups which are then referenced in the load cases and consequently load combination.</td>
</tr>
<tr>
<td>Cabled stayed Tower Example.std</td>
<td>This example shows the modeling of a tower with cable stayed and fixed at their end. The self-weight of the cables is ignored by setting the ‘Ignore Self Weight’ property of the member. After analysis for this reason of ignoring the self-weight, it will show applied self-weight is less than the total weight of all. Also, Member Tension Only attribute is set for these cables. The columns are set with standard Pipe sections whereas the cables are set with circular sections. The structure is subjected Dead and wind load in the form of lateral loads in load groups and referenced in load cases and consequently load combination.</td>
</tr>
<tr>
<td>Concrete Building Example.std</td>
<td>This physical model example shows a concrete building with steel columns. The concrete slabs are created as single physical surface for each of the floors and for the lift well with openings for the doors and windows. The slabs are subjected to distributed load categorized in the dead and live load case directly as a load item. Also, linear load combination is defined.</td>
</tr>
<tr>
<td>Steel Framed Warehouse Example.std</td>
<td>This physical model example showcases a typical warehouse with standard steel sections of various shapes. Notice that the truss chords are defined as continuous members. The members are assigned with offsets in start and end. The structure is subjected to dead and live load on the frame structure in the form of self-weight and distributed loads. Also, linear load combination is defined.</td>
</tr>
<tr>
<td>Storehouse with Walls Example.std</td>
<td>This physical model example showcases a typical storehouse with standard steel sections of various shapes. The walls of the storehouse are created as physical surfaces. The structure is subjected to dead loads on the frame structure in the form of self-weight and point loads. Also, linear load combination is defined.</td>
</tr>
</tbody>
</table>

Note: The file name of the STAAD input file is listed. Other files associated with this model are included with the same base file name in this folder using different file extensions.

EX. American Design Examples
EX. US-1 Plane Frame with Steel Design

After one analysis, member selection is requested. Since member sizes change during the member selection, another analysis is done followed by final code checking to verify that the final sizes meet the requirements of the code based on the latest analysis results.

This problem is installed with the program by default to C: \Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-1 Plane Frame with Steel Design.STD when you install the program.

where:

L = 15 ft, H1 = 20 ft, H2 = 15 ft, and H3 = 9 ft
WLx = 0.6 k/ft, WLy = 1.0 k/ft, RL = 0.9 k/ft, LL = 1.2 k/ft, P1 = 35 kips, P2 = 15 kips
Members 1, 3, & 4 are a W14x90, Members 5, 6, & 7 are a W21x50, Member 2 is a W10x49, Members 8 through 13 are a W18x35. All other members comprising the truss are a L4x4x1/4

Actual input is shown in bold lettering followed by explanation.

STAAD PLANE EXAMPLE PROBLEM NO. 1

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

UNIT FT KIP

Defines the input units for the data that follows.

JOINT COORDINATES

1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
11 15 35 ; 12 5. 38. ; 13 25 38
14 10 41 ; 15 20 41 ; 16 15 44

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCE

1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10

Defines the members by the joints to which they are connected.

MEMBER PROPERTY AMERICAN

1 3 4 TABLE ST W14X90 ; 2 TA ST W10X49
5 6 7 TA ST W21X50 ; 8 TO 13 TA ST W18X35
14 TO 23 TA ST L40404

Member properties are from the British steel table. The term ST stands for standard single section.

MEMB TRUSS

14 TO 23

The above command defines that members 14 through 23 are of type truss. This means that these members can carry only axial tension/compression and no moments.

MEMB RELEASE

5 START MZ

Member 5 has local moment-z (MZ) released at the start joint. This means that the member cannot carry any moment-z (i.e., strong axis moment) at node 3.

UNIT INCH

DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-006
DAMP 0.03

Application Examples
EX. American Design Examples
The `DEFINE MATERIAL` command initiates input for a material definition, which can later be assigned to all members. STAAD.Pro has several built-in material definitions, including the ISOTROPIC STEEL material used here. The length unit is changed from FEET to INCH to facilitate the input in familiar units.

```plaintext
CONSTANTS
BETA 90.0 MEMB 3 4
MATERIAL STEEL ALL
UNIT FT
```

The `CONSTANT` command initiates input for material constants or material definition references. The `BETA` command specifies that members 3 and 4 are rotated by 90 degrees around their own longitudinal axis. See the `4.3 Relationship Between Global and Local Coordinates` (on page 2301) for the definition of the beta angle.

A fixed support is located at joint 1 and a pinned support at joint 2.

```plaintext
SUPPORT
1 FIXED ; 2 PINNED
```

The above `PRINT` commands are self-explanatory. The `LIST` option restricts the print output to the members listed.

```plaintext
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
```

One of the components of load case 1 is the selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

```plaintext
JOINT LOAD
4 5 FY -15. ; 11 FY -35.
```

Load 1 contains joint loads also. FY indicates that the load is a force in the global Y direction.

```plaintext
MEMB LOAD
8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
```

Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The term UNI stands for uniformly distributed load. Loads are applied on members 6, and 8 to 13.

```plaintext
CALCULATE RAYLEIGH FREQUENCY
```

The above command at the end of load case 1, is an instruction to perform a natural frequency calculation based on the Rayleigh method using the data in the above load case.

```plaintext
LOADING 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 0.6 ; 8 TO 10 UNI Y -1.
```

Load case 2 is initiated and contains several member loads.

```plaintext
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
```
The above command identifies a combination load (case no. 3) with a title. The second line provides the load cases and their respective factors used for the load combination.

**Note:** Any line beginning with an asterisk (*) character is treated as a comment line.

**PERFORM ANALYSIS**

This command instructs the program to proceed with the analysis.

**LOAD LIST 1 3**

The above command activates load cases 1 and 3 only for the commands to follow. This also means that load case 2 will be made inactive.

**PRINT MEMBER FORCES**
**PRINT SUPPORT REACTION**

The above PRINT commands are self-explanatory. Also note that all the forces and reactions will be printed for load cases 1 and 3 only.

**PARAMETER**

- **CODE AISC UNIFIED**
- **NSF 0.85 ALL**
- **BEAM 1.0 ALL**
- **KY 1.2 MEMB 3 4**
- **RATIO 0.9 ALL**
- **PROFILE W14 MEMB 1 3 4**

The PARAMETER command is used to specify steel design parameters such as net section factor (NSF), effective length factor for bending about the minor axis (KY), etc. Information on these parameters can be obtained from the manual where the implementation of the code is explained. The BEAM parameter is specified to perform design at every 1/12th point along the member length, which is also the default. The RATIO parameter specifies that the ratio of actual loading over section capacity should not exceed 0.9.

**SELECT ALL**

The above command instructs the program to select the most economic section for all the members based on the results of the analysis.

**GROUP MEMB 1 3 4**
**GROUP MEMB 5 6 7**
**GROUP MEMB 8 TO 13**
**GROUP MEMB 14 TO 23**

Although the program selects the most economical section for all members, it is not always practical to use many different sizes in one structure. Grouping is a procedure by which the cross section which has the largest value for the specified attribute, which in this case is the default and hence the AREA, from among the associated member list, is assigned to all members in the list. Hence, the cross sections for members 1, 3 and 4 are replaced with the one with the largest area from among the three.

**PERFORM ANALYSIS**

As a result of the selection and grouping, the member sizes are no longer the same as the ones used in the original analysis. Hence, it is necessary to reanalyze the structure using the new properties to get new values of forces in the members.

**PARAMETER**

- **BEAM 1.0 ALL**
- **RATIO 1.0 ALL**
- **TRACK 1.0 ALL**
A new set of values are now provided for the above parameters. The actual load to member capacity \textbf{RATIO} has been redefined as 1.0. The \textbf{TRACK} parameter tells the program to print out the design results to the intermediate level of descriptively.

\begin{verbatim}
CHECK CODE ALL
\end{verbatim}

With the above command, the latest member sizes with the latest analysis results are checked to verify that they satisfy the \textbf{CODE} specifications.

\begin{verbatim}
STEEL TAKE OFF
\end{verbatim}

This command instructs the program to list the length and weight of all the different member sizes.

\begin{verbatim}
FINISH
\end{verbatim}

This command terminates the STAAD run.

\section*{Input File}

\begin{verbatim}
STAAD PLANE EXAMPLE PROBLEM NO. 1
UNIT FT KIP
JOINT COORDINATES
  1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
  7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
  11 15 35 ; 12 5. 38. ; 13 25 38
  14 10 41 ; 15 20 41 ; 16 15 44
MEMBER INCIDENCE
  1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
  6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
  10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
  14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
  18 10 15 ; 19 10 13 ; 20 7 9
  21 9 11 ; 22 10 11 ; 23 8 10
MEMBER PROPERTY AMERICAN
  1 3 4 TA ST W14X90 ; 2 TA ST W10X49
  5 6 7 TA ST W21X50 ; 8 TO 13 TA ST W18X35
  14 TO 23 TA ST L40404
*MEMB TRUSS
*MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99
UNIT INCH
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 29000
  POISSON 0.3
  DENSITY 283e-006
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
  END DEFINE MATERIAL
  CONSTANTS
  MATERIAL STEEL ALL
  BETA 90.0 MEMB 3 4
  UNIT FT
  SUPPORT
  1 FIXED ; 2 PINNED
\end{verbatim}
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTY LIST 1 2 5 8 14
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -15. ; 11 FY -35.
MEMB LOAD
8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
CALCULATE RAYLEIGH FREQUENCY
LOADING 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 0.6 ; 8 TO 10 UNI Y -1.
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
PARAMETER
CODE AISC UNIFIED
NSF 0.85 ALL
BEAM 1.0 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
PROFILE W14 MEMB 1 3 4
SELECT ALL
GROUP MEMB 1 3 4
GROUP MEMB 5 6 7
GROUP MEMB 8 TO 13
GROUP MEMB 14 TO 23
PERFORM ANALYSIS
PARAMETER
BEAM 1.0 ALL
RATIO 1.0 ALL
TRACK 1.0 ALL
CHECK CODE ALL
STEEL TAKE OFF
FINISH

STAAD Output File

*****************************************************************************************************************************
*                                                                                                                          *
*           STAAD.Pro CONNECT Edition                                                                                       *
*           Version  22.01.00.**                                                                                           *
*           Proprietary Program of                                                                                         *
*           Bentley Systems, Inc.                                                                                           *
*           Date=    APR 14, 2019                                                                                           *
*           Time=    22:55: 5                                                                                               *
*                                                                                                                          *
* Licensed to: Bentley Systems Inc                                                                                         *
*****************************************************************************************************************************

1. STAAD PLANE EXAMPLE PROBLEM NO. 1
INPUT FILE: US-1 Plane Frame with Steel Design.STD
2. UNIT FT KIP
3. JOINT COORDINATES
4. 1 0 0 ; 2 30 0 ; 3 0 20 0 6 30 20 0
5. 7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
6. 11 15 35 ; 12 5 38. ; 13 25 38
7. 14 10 41 ; 15 20 41 ; 16 15 44
8. MEMBER INCIDENCE
9. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
10. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
11. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
12. 14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
13. 18 10 15 ; 19 10 13 ; 20 7 9
14. 21 9 11 ; 22 10 11 ; 23 8 10
15. MEMBER PROPERTY AMERICAN
16. 1 3 4 TA ST W14X90 ; 2 TA ST W10X49
17. 5 6 7 TA ST W21X50 ; 8 TO 13 TA ST W18X35
18. 14 TO 23 TA ST L40X40
19. *MEMB TRUSS
20. *14 TO 23
21. MEMB RELEASE
22. 5 START MZ
23. 14 TO 23 START MPY 0.99 MPZ 0.99
24. 14 TO 23 END MPY 0.99 MPZ 0.99
25. UNIT INCH
26. DEFINE MATERIAL START
27. ISOTROPIC STEEL
28. E 29000
29. POISSON 0.3
30. DENSITY 283E-006
31. ALPHA 6E-006
32. DAMP 0.03
33. TYPE STEEL
34. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
35. END DEFINE MATERIAL
36. CONSTANTS
37. MATERIAL STEEL ALL
38. BETA 90.0 MEMB 3 4
39. EXAMPLE PROBLEM NO. 1
40. UNIT FT
41. SUPPORT
42. PRINT MEMBER INFORMATION LIST 1 5 14
MEMBER INFORMAT LIST 1
EXAMPLE PROBLEM NO. 1
43. PRINT MEMBER PROPERTY LIST 1 2 5 8 14
MEMBER PROPERTY LIST 1
EXAMPLE PROBLEM NO. 1
44. UNIT INCH
MEMBER PROPERTIES. UNIT - INCH
MEMBER START END LENGTH BETA
JOINT JOINT (FEET) (DEG) RELEASES
1 1 3 20.000 0.00
5 3 4 10.000 0.00 000001000000
14 9 12 3.905 0.00 000011000011
*************** END OF DATA FROM INTERNAL STORAGE ***************
43. PRINT MEMBER PROPERTY LIST 1 2 5 8 14
MEMBER PROPERTY LIST 1
EXAMPLE PROBLEM NO. 1
MEMBER PROPERTIES. UNIT - INCH
MEMBER PROFILE AX/ IZ/ IY/ IX/
AY AZ SZ SY
1 ST W14X90 26.50 999.00 362.00 4.06
6.16  13.73  142.71  49.93
2 ST W10X49  14.40  272.00  93.40  1.39
3.40  7.47  54.40  18.68
5 ST W21X50  14.70  984.00  24.90  1.14
7.90  4.66  94.62  7.63
8 ST W18X35 10.30  510.00  15.30  0.51
5.31  3.40  57.63  5.10
14 ST L40404 1.93  1.18  4.90  0.04
0.67  0.67  0.76  1.73

************ END OF DATA FROM INTERNAL STORAGE ************

44. LOADING 1 DEAD AND LIVE LOAD
45. SELFWEIGHT Y -1.0
46. JOINT LOAD
47. 4 5 FY -15. ; 11 FY -35.
48. MEMB LOAD
49. 8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
50. CALCULATE RAYLEIGH FREQUENCY
51. LOADING 2 WIND FROM LEFT
52. MEMBER LOAD
53. 1 2 UNI GX 0.6 ; 8 TO 10 UNI Y -1.
54. * 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
55. LOAD COMB 3 75 PERCENT DL LL WL
56. 1 0.75 2 0.75
57. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS         16  NUMBER OF MEMBERS      23
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
EXAMPLE PROBLEM NO. 1 -- PAGE NO. 5
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     43
TOTAL LOAD COMBINATION CASES =     1  SO FAR.
**********************************************************
*                                                        *
* RAYLEIGH FREQUENCY FOR LOADING     1 =    3.13858 CPS  *
* MAX DEFLECTION =  1.21777 INCH GLO X, AT JOINT     7  *
*                                                        *
**********************************************************

58. LOAD LIST 1 3
59. PRINT MEMBER FORCES

MEMBER FORCES
EXAMPLE PROBLEM NO. 1 -- PAGE NO. 6
MEMBER END FORCES  STRUCTURE TYPE = PLANE

All units are -- KIP FEET (LOCAL)

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>54.05</td>
<td>-2.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-61.74</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-52.25</td>
<td>2.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>21.74</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>40.71</td>
<td>18.99</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>247.88</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-39.36</td>
<td>-9.99</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>41.97</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>33.81</td>
<td>-5.48</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-21.74</td>
<td></td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>-33.07</td>
<td>5.48</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-60.47</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>28.90</td>
<td>-0.16</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-41.97</td>
<td></td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>-28.35</td>
<td>6.91</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-11.09</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>58.79</td>
<td>0.00</td>
<td>-2.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>-56.99</td>
<td>0.00</td>
<td>2.00</td>
<td>0.00</td>
<td>40.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>2</td>
<td>55.17</td>
<td>0.00</td>
<td>-3.51</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>31.94</td>
<td>0.00</td>
<td>-5.48</td>
<td>0.00</td>
<td>23.19</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-30.59</td>
<td>0.00</td>
<td>5.48</td>
<td>0.00</td>
<td>105.29</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-30.65</td>
<td>0.00</td>
<td>13.66</td>
<td>0.00</td>
<td>99.65</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**EXAMPLE PROBLEM NO. 1**

MEMBER END FORCES  STRUCTURE TYPE = PLANE

**ALL UNITS ARE -- KIP  FEET  (LOCAL )**
EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    8
MEMBER END FORCES    STRUCTURE TYPE = PLANE
-----------------
ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
3    10     5.69      0.01     0.00      0.00      0.00      -0.00
15     -5.66      0.01     0.00      0.00      0.00       0.00
19    10     1.97      0.01     0.00      0.00      0.00      -0.00
13     -1.95      0.01     0.00      0.00      0.00       0.00
3    10     -6.90      0.01     0.00      0.00      0.00       0.00
13      6.91      0.01     0.00      0.00      0.00       0.00
20    1     7   -17.29      0.02     0.00      0.00      0.00       0.00
  9     17.29      0.02     0.00      0.00      0.00       0.00
  3    19.94      0.02     0.00      0.00      0.00       0.00
  9     19.94      0.02     0.00      0.00      0.00       0.00
21    1     9   -19.44      0.02     0.00      0.00      0.00       0.00
 11     19.44      0.02     0.00      0.00      0.00       0.00
  3    14.52      0.02     0.00      0.00      0.00       0.00
 11     14.52      0.02     0.00      0.00      0.00       0.00
22    1     10   -21.48      0.02     0.00      0.00      0.00       0.00
 11     21.48      0.02     0.00      0.00      0.00       0.00
  3     -5.75      0.02     0.00      0.00      0.00       0.00
 11     5.75      0.02     0.00      0.00      0.00       0.00
23    1     8   -23.40      0.02     0.00      0.00      0.00       0.00
 10     23.40      0.02     0.00      0.00      0.00       0.00
  3     0.86      0.02     0.00      0.00      0.00       0.00
 10     -0.86      0.02     0.00      0.00      0.00       0.00
************** END OF LATEST ANALYSIS RESULT **************

60. PRINT SUPPORT REACTION
SUPPORT REACTION
EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    9
SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
-----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
 1    1      2.00     54.05      0.00      0.00      0.00    -61.74
 3    1     -2.00     58.79      0.00      0.00      0.00      0.00
************** END OF LATEST ANALYSIS RESULT **************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

***NOTE : AISC 360-16 Design Statement for STAAD.Pro.

*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING: -

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that
produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase
based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being ◊about◊ the relevant axis (X/Y), while
the results for shear are reported as being for shear forces ◊along◊ the axis.
   E.g : Mx indicates bending about the X axis, while Vx indicates shear along
the X axis.

*** ABBREVIATIONS ***:

F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
F-L-B = Flange Local Buckling
W-L-B = Web Local Buckling
L-L-B = Leg Local Buckling
C-F-Y = Compression Flange Yielding
T-F-Y = Tension Flange Yielding

EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   11
STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 1

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile: ST W14X68</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status: PASS</td>
<td>Ratio: 0.833</td>
<td>Loadcase: 3</td>
</tr>
<tr>
<td>Location: 0.00</td>
<td>Ref: Eq.H1-3a(H1-1b)</td>
<td></td>
</tr>
<tr>
<td>Pz: 40.71 C</td>
<td>Vy: 18.99</td>
<td>Vx: 0.00</td>
</tr>
<tr>
<td>Tz: 0.000</td>
<td>My: 0.000</td>
<td>Mx: 247.9</td>
</tr>
</tbody>
</table>

- Member : 2

---

STAAD.Pro 4323 User Manual
Member No:        2  Profile:  ST  W12X26  (AISC SECTIONS)  
Status:        PASS  Ratio:         0.675  Loadcase:        1  
Location:  15.00  Ref:     Eq.H1-3a(H1-1b)  
Pz:       33.07  C  Vy:        -5.481  Vx:       0.000  
Tz:       0.000  My:        0.000  Mx:       60.47  

- Member : 3

Member No:        3  Profile:  ST  W14X61  (AISC SECTIONS)  
Status:        PASS  Ratio:         0.888  Loadcase:        3  
Location:  20.00  Ref:     Eq.H1-1b  
Pz:       53.82  C  Vy:        0.000  Vx:      -3.507  
Tz:       0.000  My:      105.3  Mx:       0.000  

- Member : 4

Member No:        4  Profile:  ST  W14X90  (AISC SECTIONS)  
Status:        PASS  Ratio:         0.538  Loadcase:        3  
Location:  0.00  Ref:     Eq.H1-1b  
Pz:       31.66  T  Vy:       -13.66  Vx:       0.000  
Tz:       0.000  My:       105.3  Mx:      -185.4  

- Member : 5

Member No:        5  Profile:  ST  W18X40  (AISC SECTIONS)  
Status:        PASS  Ratio:         0.864  Loadcase:        1  
Location:  10.00  Ref:     Eq.H1-1b  
Pz:       3.481  T  Vy:        17.95  Vx:       0.000  
Tz:       0.000  My:       105.3  Mx:      -182.0  

- Member : 6

Member No:        6  Profile:  ST  W21X44  (AISC SECTIONS)  
Status:        PASS  Ratio:         0.842  Loadcase:        1  
Location:  2.50  Ref:     Eq.H1-1b  
Pz:       3.481  T  Vy:       -17.61  Vx:       0.000  
Tz:       0.000  My:       105.3  Mx:      -185.4  

- Member : 7

Member No:        7  Profile:  ST  W18X40  (AISC SECTIONS)  
Status:        PASS  Ratio:         0.842  Loadcase:        3  
Location:  10.00  Ref:     Eq.H1-1b  
Pz:       10.16  T  Vy:       -22.16  Vx:       0.000  
Tz:       0.000  My:       105.3  Mx:       175.4  

example problem no. 1  -- page no. 12  
staad.pro member selection - aisc 360-16 lrfd (v1.1)  
******************************************************************************
all units are - kip feet (unless otherwise noted).  
- member : 8

Member No:        8  Profile:  ST  W14X22  (AISC SECTIONS)  
Status:        PASS  Ratio:         0.771  Loadcase:        1  
Location:  0.00  Ref:     Eq.H1-3a(H1-1b)  
Pz:       36.53  C  Vy:        16.62  Vx:       0.000  
Tz:       0.000  My:       105.3  Mx:       60.47  

staad.pro 4324 user manual
<table>
<thead>
<tr>
<th>Member</th>
<th>Member No.</th>
<th>Profile</th>
<th>Status</th>
<th>Ratio</th>
<th>Loadcase</th>
<th>Location</th>
<th>Ref</th>
<th>Pz</th>
<th>Vx</th>
<th>Vy</th>
<th>My</th>
<th>Mx</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>9</td>
<td>ST W12X22</td>
<td>PASS</td>
<td>0.809</td>
<td>1</td>
<td>5.83</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>36.66</td>
<td>0.000</td>
<td>3.515</td>
<td>0.000</td>
<td>-56.94</td>
</tr>
<tr>
<td>10</td>
<td>10</td>
<td>ST W12X26</td>
<td>PASS</td>
<td>0.816</td>
<td>1</td>
<td>5.83</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>41.74</td>
<td>0.000</td>
<td>-25.02</td>
<td>0.000</td>
<td>73.14</td>
</tr>
<tr>
<td>11</td>
<td>11</td>
<td>ST W12X26</td>
<td>PASS</td>
<td>0.816</td>
<td>1</td>
<td>5.83</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>41.72</td>
<td>0.000</td>
<td>-25.06</td>
<td>0.000</td>
<td>73.14</td>
</tr>
<tr>
<td>12</td>
<td>12</td>
<td>ST W14X22</td>
<td>PASS</td>
<td>0.777</td>
<td>1</td>
<td>5.83</td>
<td>Eq.H1-3a(H1-1a)</td>
<td>39.97</td>
<td>0.000</td>
<td>2.432</td>
<td>0.000</td>
<td>-57.15</td>
</tr>
<tr>
<td>13</td>
<td>13</td>
<td>ST W14X30</td>
<td>PASS</td>
<td>0.829</td>
<td>3</td>
<td>0.00</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>26.74</td>
<td>0.000</td>
<td>19.68</td>
<td>0.000</td>
<td>99.65</td>
</tr>
<tr>
<td>14</td>
<td>14</td>
<td>ST L20X20</td>
<td>PASS</td>
<td>0.754</td>
<td>3</td>
<td>0.00</td>
<td>Eq.H2-1</td>
<td>5.625</td>
<td>0.000</td>
<td>-4554E-02</td>
<td>-1012E-03</td>
<td>1012E-03</td>
</tr>
<tr>
<td>15</td>
<td>15</td>
<td>ST L25X25</td>
<td>PASS</td>
<td>0.235</td>
<td>1</td>
<td>0.00</td>
<td>C13</td>
<td>1.824</td>
<td>0.000</td>
<td>-5918E-02</td>
<td>-5918E-02</td>
<td></td>
</tr>
</tbody>
</table>
### Member Selection

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST L30253 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.821</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>3.91 Ref: Eq.H2-1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>24.43 T Vy: 0.2114E-03 Vx: 0.2114E-03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000 My: 0.2237E-01 Mx: -0.2237E-01</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST L20204 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.877</th>
<th>Loadcase: 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>3.91 Ref: Eq.H2-1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>23.99 T Vy: 0.1653E-03 Vx: 0.1653E-03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000 My: 0.1677E-01 Mx: -0.1677E-01</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST L25253 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.733</th>
<th>Loadcase: 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>0.00 Ref: Cl.E3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>5.694 C Vy: -0.4424E-02 Vx: -0.4424E-02</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000 My: 0.7830E-04 Mx: -0.7830E-04</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST L20202 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.804</th>
<th>Loadcase: 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>2.28 Ref: Eq.H2-1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>6.907 T Vy: 0.4779E-03 Vx: 0.4779E-03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000 My: 0.4374E-02 Mx: -0.4374E-02</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST L25253 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.790</th>
<th>Loadcase: 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>3.75 Ref: Eq.H2-1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>19.94 T Vy: -0.2987E-04 Vx: -0.2987E-04</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000 My: 0.2461E-01 Mx: -0.2461E-01</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST L25253 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.785</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>3.75 Ref: Eq.H2-1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>19.44 T Vy: -0.1450E-03 Vx: -0.1450E-03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000 My: 0.3268E-01 Mx: -0.3268E-01</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**EXAMPLE PROBLEM NO. 1**

---

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).
69. GROUP MEMB 1 3 4
GROUPING BASED ON MEMBER
4 (ST W14X90 ) LIST= 1....
70. GROUP MEMB 5 6 7
GROUPING BASED ON MEMBER
6 (ST W21X44 ) LIST= 5....
71. GROUP MEMB 8 TO 13
GROUPING BASED ON MEMBER
13 (ST W14X30 ) LIST= 8....
72. GROUP MEMB 14 TO 23
GROUPING BASED ON MEMBER
23 (ST L30253 ) LIST= 14....
73. PERFORM ANALYSIS
** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
****************************************************************************
** RAYLEIGH FREQUENCY FOR LOADING 1 = 2.89617 CPS **
** MAX DEFLECTION = 1.54692 INCH GLO X, AT JOINT 7 **
****************************************************************************
74. PARAMETER
75. BEAM 1.0 ALL
76. RATIO 1.0 ALL
77. TRACK 1.0 ALL
78. CHECK CODE ALL
STEEL DESIGN
EXAMPLE PROBLEM NO. 1
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
****************************************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).
**NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:
The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING: -
<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
</tbody>
</table>
### Torsional moment.

### Bending moment about minor axis.

### Bending moment about major axis.

---

### DESIGN MESSAGES:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.

2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.

3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis.

**E.g**: Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

---

### ABBREVIATIONS:

- **F-T-B** = Flexural-Torsional Buckling
- **L-T-B** = Lateral-Torsional Buckling
- **F-L-B** = Flange Local Buckling
- **W-L-B** = Web Local Buckling
- **L-L-B** = Leg Local Buckling
- **C-F-Y** = Compression Flange Yielding
- **T-F-Y** = Tension Flange Yielding

---

### EXAMPLE PROBLEM NO. 1

#### STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (VI.1)

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE NOTED).**

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile:</th>
<th>ST W14X90</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>PASS</td>
<td>Ratio: 0.625</td>
<td>Loadcase: 3</td>
</tr>
<tr>
<td>Location:</td>
<td>0.00</td>
<td>Ref: Eq.H1-3a(H1-1b)</td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>40.46 C</td>
<td>Vy: 19.00</td>
<td>Vx: 0.000</td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000 My: 0.000</td>
<td>Mx: 254.3</td>
<td></td>
</tr>
</tbody>
</table>

---

### SLENDERNESS

**Actual Slenderness Ratio**: 64.935

**Allowable Slenderness Ratio**: 200.000

**LOC**: 0.00

---

### STRENGTH CHECKS

**Critical L/C**: 3

**Ratio**: 0.625(PASS)

**Condition**: Eq.H1-3a(H1-1b)

---

### SECTION PROPERTIES

<table>
<thead>
<tr>
<th>Ag: 2.650E+01</th>
<th>Axx: 2.059E+01</th>
<th>Ayy: 6.160E+00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ixx: 9.990E+02</td>
<td>Iyy: 3.620E+02</td>
<td>J: 4.060E+00</td>
</tr>
<tr>
<td>Sxx+: 1.427E+02</td>
<td>Sxx-: 1.427E+02</td>
<td>Zxx: 1.570E+02</td>
</tr>
<tr>
<td>Syy+: 4.993E+01</td>
<td>Syy-: 4.993E+01</td>
<td>Zyy: 7.560E+01</td>
</tr>
<tr>
<td>Cw: 1.593E+04</td>
<td>x0: 0.000E+00</td>
<td>y0: 0.000E+00</td>
</tr>
</tbody>
</table>

### MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Fyld: 5184.000</th>
<th>Fu: 8351.999</th>
</tr>
</thead>
</table>

- **Actual Member Length**: 20.000

### Design Parameters

- **Kx**: 1.00
- **Ky**: 1.00
- **NSF**: 0.85
- **SLF**: 1.00
- **CSP**: 1.00

### COMPRESSION CLASSIFICATION

- **L/C**: 1
- **LOC**: 0.00
Application Examples
EX. American Design Examples

| Flange: NonSlender | 10.21 | N/A | 15.89 | Table.4.1a.Case1 |
| Web: NonSlender   | 28.59 | N/A | 42.29 | Table.4.1a.Case5 |
| Web: Compact      | 28.59 | 106.72 | 161.78 | Table.4.1b.Case15 |

**FLEXURE CLASSIFICATION**

| Flange: Compact | 10.21 | 10.79 | 28.38 | Table.4.1b.Case10 |
| Web: Compact    | 28.59 | 106.72 | 161.78 | Table.4.1b.Case15 |

**EXAMPLE PROBLEM NO. 1**

*STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)*

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

- Member: 1 Contd.

**CHECKS FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>858.6</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>858.6</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECKS FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>53.12</td>
<td>792.2</td>
<td>0.067</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>53.12</td>
<td>687.7</td>
<td>0.077</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>53.12</td>
<td>728.1</td>
<td>0.073</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECKS FOR SHEAR**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.000</td>
<td>400.3</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>19.00</td>
<td>133.1</td>
<td>0.143</td>
<td>Cl.G1</td>
<td>3</td>
</tr>
</tbody>
</table>

**CHECKS FOR BENDING**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>254.3</td>
<td>423.9</td>
<td>0.600</td>
<td>Cl.F2.1</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>204.1</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt X</td>
<td>254.3</td>
<td>423.9</td>
<td>0.600</td>
<td>Cl.F2.2</td>
<td>3</td>
</tr>
</tbody>
</table>

**CHECKS FOR AXIAL BEND INTERACTION**

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H1</td>
<td>0.629</td>
<td>Eq.H1-1b</td>
<td>3</td>
</tr>
<tr>
<td>Cl.H13-IP</td>
<td>0.625</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>3</td>
</tr>
<tr>
<td>Cl.H13-QP</td>
<td>0.178</td>
<td>Eq.H1-3b</td>
<td>3</td>
</tr>
</tbody>
</table>
**Member No:** 2  |  **Profile:** ST W12X26  |  (AISC SECTIONS)
**Status:** PASS  |  Ratio: 0.712  |  **Loadcase:** 1
**Location:** 15.00  |  **Ref:** Eq.H1-3a(H1-1b)
**Pz:** 32.75  |  **C**  |  **Vy:** -6.201  |  **Vx:** 0.000
**Tz:** 0.000  |  **My:** 0.000  |  **Mx:** 64.21

**SLENDERNESS**
Actual Slenderness Ratio: 119.696
Allowable Slenderness Ratio: 200.000  |  **LOC:** 0.00

**STRENGTH CHECKS**
Critical L/C: 1  |  Ratio: 0.712 (PASS)
Loc: 15.00  |  **Condition:** Eq.H1-3a(H1-1b)

**SECTION PROPERTIES**  
**Ag:** 7.650E+00  |  **Axx:** 4.932E+00  |  **Ayy:** 2.806E+00
**Ixx:** 2.040E+02  |  **Iyy:** 1.730E+01  |  **J:** 3.000E-01
**Sxx+:** 3.344E+01  |  **Sxx-:** 3.344E+01  |  **Zxx:** 3.720E+01
**Syy+:** 5.331E+00  |  **Syy-:** 5.331E+00  |  **Zyy:** 8.170E+00
**Cw:** 2.079E+03  |  **x0:** 0.000E+00  |  **y0:** -8.882E-16

**MATERIAL PROPERTIES**
**Fyld:** 5184.000  |  **Fu:** 8351.999

Actual Member Length: 15.000
Design Parameters
**Kx:** 1.00  |  **Ky:** 1.00  |  **NSF:** 0.85  |  **SLF:** 1.00  |  **CSP:** 1.00

**COMPRESSION CLASSIFICATION**
**Flange:** NonSlender  |  8.54  |  **N/A:** 15.89  |  Table.4.1a.Case1
**Web:** Slender  |  49.74  |  **N/A:** 42.29  |  Table.4.1a.Case5

**FLEXURE CLASSIFICATION**
**Flange:** Compact  |  8.54  |  10.79  |  28.38  |  Table.4.1b.Case10
**Web:** Compact  |  49.74  |  106.72  |  161.78  |  Table.4.1b.Case15

---

**EXAMPLE PROBLEM NO. 1**
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

---

### CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>247.9</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>247.9</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

---

### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>33.14</td>
<td>225.4</td>
<td>0.147</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>33.14</td>
<td>116.6</td>
<td>0.284</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Member No:</td>
<td>3</td>
<td>Profile: ST W14X90</td>
<td>(AISC SECTIONS)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-----------</td>
<td>---</td>
<td>-------------------</td>
<td>-----------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Status:</td>
<td>PASS</td>
<td>Ratio: 0.386</td>
<td>Loadcase: 3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Location:</td>
<td>20.00</td>
<td>Ref: Eq.H1-1b</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:</td>
<td>53.40</td>
<td>C</td>
<td>Vy: 0.000</td>
<td>Vx: -3.505</td>
<td></td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td></td>
<td>My: -70.09</td>
<td>Mx: 0.000</td>
<td></td>
</tr>
</tbody>
</table>

**SLENDERNESS**
- Actual Slenderness Ratio: 77.922
- Allowable Slenderness Ratio: 200.000
- LOC: 0.00

**STRENGTH CHECKS**
- Critical L/C: 3
- Ratio: 0.386(PASS)
- Condition: Eq.H1-1b

**SECTION PROPERTIES**
- (LOC: 20.00, PROPERTIES UNIT: IN )
- Ag: 2.658E+01
- Axx: 2.059E+01
- Ayy: 3.620E+02
- J: 4.060E+00
- Sxx+: 1.427E+02
- Sxx−: 1.427E+02
- Zxx: 1.570E+02
- Syy+: 4.993E+01
- Zyy: 7.560E+01
- Cw: 1.593E+04
- x0: 0.000E+00
- y0: 0.000E+00

**MATERIAL PROPERTIES**
- Fyld: 5184.000
- Fu: 8351.999

**Actual Member Length:** 20.00

**Design Parameters**
- (Rolled)
- Kx: 1.00
- Ky: 1.20
- NSF: 0.85
- SLF: 1.00
- CSP: 1.00
### COMPRESSION CLASSIFICATION (L/C: 1 LOC: 0.00)

<table>
<thead>
<tr>
<th>Flange: NonSlender</th>
<th>10.21</th>
<th>N/A</th>
<th>15.89</th>
<th>Table.4.1a.Case1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Web: NonSlender</td>
<td>28.59</td>
<td>N/A</td>
<td>42.29</td>
<td>Table.4.1a.Case5</td>
</tr>
</tbody>
</table>

### FLEXURE CLASSIFICATION (L/C: 1 LOC: 0.00)

<table>
<thead>
<tr>
<th>Flange: Compact</th>
<th>10.21</th>
<th>10.79</th>
<th>28.38</th>
<th>Table.4.1b.Case13</th>
</tr>
</thead>
<tbody>
<tr>
<td>Web: Compact</td>
<td>28.59</td>
<td>*****</td>
<td>*****</td>
<td>Table.4.1.</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Demands</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>858.6</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1 0.00</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>858.6</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1 0.00</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Demands</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>58.83</td>
<td>792.2</td>
<td>0.074</td>
<td>Cl.E3</td>
<td>1 0.00</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>58.83</td>
<td>623.7</td>
<td>0.094</td>
<td>Cl.E3</td>
<td>1 0.00</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>58.83</td>
<td>728.1</td>
<td>0.081</td>
<td>Cl.E4</td>
<td>1 0.00</td>
</tr>
</tbody>
</table>

### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>Demands</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>3.505</td>
<td>400.3</td>
<td>0.009</td>
<td>Cl.G1</td>
<td>3 0.00</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>133.1</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1 0.00</td>
</tr>
</tbody>
</table>

### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>Demands</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>0.000</td>
<td>423.9</td>
<td>0.000</td>
<td>Cl.F2.1</td>
<td>1 0.00</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>70.09</td>
<td>204.1</td>
<td>0.343</td>
<td>Cl.F6.1</td>
<td>3 20.00</td>
</tr>
<tr>
<td>L-T-B Abt X</td>
<td>0.000</td>
<td>406.2</td>
<td>0.000</td>
<td>Cl.F2.2</td>
<td>1 0.00</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.386</td>
<td>Eq.H1-1b</td>
<td>3</td>
<td>20.00</td>
</tr>
</tbody>
</table>
**Member No:** 4  **Profile:** ST W14X90  **(AISC SECTIONS)**

| Status: PASS | Ratio: 0.556 | Loadcase: 3 |

**Location:** 0.00  **Ref:** Eq.H1-1b

| Pz: 31.18 | C | Vy: 0.000 | Vx: -13.76 |
| Tz: 0.000 | My: 109.0 | Mx: 0.000 |

**SLENDERNESS**

- **Actual Slenderness Ratio:** 58.442
- **Allowable Slenderness Ratio:** 200.000  **LOC:** 0.00

**STRENGTH CHECKS**

- **Critical L/C:** 3  **Ratio:** 0.556(PASS)  **Condition:** Eq.H1-1b

**SECTION PROPERTIES**

- **(LOC: 0.00, PROPERTIES UNIT: IN )**
  - Ag: 2.650E+01  **Axx:** 2.059E+01  **Ayy:** 6.160E+00
  - **Ixx:** 9.990E+02  **Iyy:** 3.620E+02  **J:** 4.060E+00
  - **Sxx+:** 1.427E+02  **Sxx-:** 1.427E+02  **Zxx:** 1.570E+02
  - **Syy+:** 4.993E+01  **Syy-:** 4.993E+01  **Zyy:** 7.560E+01
  - **Cw:** 1.593E+04  **x0:** 0.000E+00  **y0:** 0.000E+00

**MATERIAL PROPERTIES**

- **Fyld:** 5184.000  **Fu:** 8351.999

**Actual Member Length:** 15.000

**Design Parameters**  **(Rolled)**

- **Kx:** 1.00  **Ky:** 1.20  **NSF:** 0.85  **SLF:** 1.00  **CSP:** 1.00

**COMPRESSION CLASSIFICATION**

- **(L/C: 1 LOC: 0.00)**
  - Flange: NonSlender 10.21  **N/A**  15.89  **Table.4.1a.Case1**
  - Web: NonSlender 28.59  **N/A**  42.29  **Table.4.1a.Case5**

**FLEXURE CLASSIFICATION**

- **(L/C: 1 LOC: 0.00)**
  - Flange: Compact 10.21  **10.79**  28.38  **Table.4.1b.Case13**
  - Web: Compact 28.59  **********  **Table.4.1.**

**EXAMPLE PROBLEM NO. 1**  -- PAGE NO. 23

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

<table>
<thead>
<tr>
<th>CHECKS FOR AXIAL TENSION</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DEMAND</strong></td>
</tr>
<tr>
<td>Ten. Yld.</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
</tr>
</tbody>
</table>

**CHECKS FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th><strong>DEMAND</strong></th>
<th><strong>CAPACITY</strong></th>
<th><strong>RATIO</strong></th>
<th><strong>REFERENCE</strong></th>
<th><strong>L/C</strong></th>
<th><strong>LOC</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>31.88</td>
<td>820.6</td>
<td>0.039</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>31.88</td>
<td>717.3</td>
<td>0.044</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
</tbody>
</table>
### Checks for Shear

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>13.76</td>
<td>400.3</td>
<td>0.034</td>
<td>Cl.G1</td>
<td>3</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.000</td>
<td>133.1</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

### Checks for Bending

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>0.000</td>
<td>423.9</td>
<td>0.000</td>
<td>Cl.F2.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>-109.0</td>
<td>204.1</td>
<td>0.534</td>
<td>Cl.F6.1</td>
<td>3</td>
</tr>
</tbody>
</table>

### Checks for Axial Bend Interaction

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H1</td>
<td>0.556</td>
<td>Eq.H1-1b</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**Example Problem No. 1**  
--- Page No. 24

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE NOTED).**
- Member: 5

---

### SLENDERNESS

- Actual Slenderness Ratio: 95.097
- Allowable Slenderness Ratio: 300.000
- LOC: 0.00

### Strength Checks

- Critical L/C: 1  
- Ratio: 0.702 (PASS)
- Location: 10.00  
- Ref: Eq.H1-1b

### Section Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>1.300E+01</td>
</tr>
<tr>
<td>Axx</td>
<td>5.850E+00</td>
</tr>
<tr>
<td>Ayy</td>
<td>7.245E+00</td>
</tr>
<tr>
<td>Ixx</td>
<td>8.430E+02</td>
</tr>
<tr>
<td>Iyy</td>
<td>2.070E+01</td>
</tr>
<tr>
<td>J</td>
<td>7.700E-01</td>
</tr>
<tr>
<td>Sxx+</td>
<td>8.145E+01</td>
</tr>
<tr>
<td>Sxx-</td>
<td>8.145E+01</td>
</tr>
<tr>
<td>Syy+</td>
<td>6.369E+00</td>
</tr>
<tr>
<td>Syy-</td>
<td>6.369E+00</td>
</tr>
<tr>
<td>Zxx</td>
<td>9.540E+01</td>
</tr>
<tr>
<td>Zyy</td>
<td>1.020E+01</td>
</tr>
<tr>
<td>Cw</td>
<td>2.540E+03</td>
</tr>
<tr>
<td>x0</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>y0</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>Mx</td>
<td>-179.5</td>
</tr>
<tr>
<td>My</td>
<td>0.000</td>
</tr>
<tr>
<td>Vy</td>
<td>17.73</td>
</tr>
<tr>
<td>Vx</td>
<td>0.000</td>
</tr>
</tbody>
</table>

### Material Properties

- Fyld: 5184.000
- Fu: 8351.999

---

**Actual Member Length:** 10.000  
**Design Parameters:**
- Kx: 1.00
- Ky: 1.00
- NSF: 0.85
- SLF: 1.00
- CSP: 1.00

---

**Compression Classification:**
- L/C: 1
- LOC: 120.00
### Flange: NonSlender       7.22       N/A      15.89     Table.4.1a.Case1
### Web   : Slender         56.57       N/A      42.29     Table.4.1a.Case5

**FLEXURE CLASSIFICATION**  
(L/C: 1 LOC: 120.00)  
\[ 1 \quad l \quad l \quad p \quad l \quad r \quad CASE \]

### Flange: Compact          7.22      10.79     28.38     Table.4.1b.Case10
### Web   : Compact         56.57     106.72    161.78     Table.4.1b.Case15

---

### EXAMPLE PROBLEM NO. 1  
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)  
**************************************************  
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).  
- Member : 5 Contd.

#### CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>10.25</td>
<td>421.2</td>
<td>0.024</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>10.25</td>
<td>421.2</td>
<td>0.024</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
</tbody>
</table>

#### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>0.000</td>
<td>374.0</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>0.000</td>
<td>256.6</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>0.000</td>
<td>316.0</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

#### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.000</td>
<td>113.7</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>18.17</td>
<td>156.5</td>
<td>0.116</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

#### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>-179.5</td>
<td>257.6</td>
<td>0.697</td>
<td>Cl.F2.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>27.52</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt X</td>
<td>-179.5</td>
<td>257.6</td>
<td>0.697</td>
<td>Cl.F2.2</td>
<td>1</td>
</tr>
</tbody>
</table>

#### CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H1</td>
<td>0.702</td>
<td>Eq.H1-1b</td>
<td>1</td>
</tr>
</tbody>
</table>

---

### EXAMPLE PROBLEM NO. 1  
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)  
**************************************************  
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).  
- Member : 6

<table>
<thead>
<tr>
<th>Member No:</th>
<th>6</th>
<th>Profile: ST W21X44</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>PASS</td>
<td>Ratio: 0.827</td>
<td>Loadcase: 1</td>
</tr>
</tbody>
</table>
Location: 2.50  Ref: Eq.H1-1b
Pz: 4.078  T  Vy: -.3776  Vx: 0.000
Tz: 0.000  My: 0.000  Mx: -182.5

SLENDERNESS
Actual Slenderness Ratio: 95.097
Allowable Slenderness Ratio: 300.000  LOC: 0.00

STRENGTH CHECKS
Critical L/C: 1  Ratio: 0.827 (PASS)
Loc: 2.50  Condition: Eq.H1-1b

SECTION PROPERTIES (LOC: 2.50, PROPERTIES UNIT: IN)
Ag: 1.300E+01  Axx: 5.850E+00  Ayy: 7.245E+00
Ixx: 8.430E+02  Iyy: 2.070E+01  J: 7.700E-01
Sxx+: 8.145E+01  Sxx-: 8.145E+01  Zxx: 9.540E+01
Syy+: 6.369E+00  Syy-: 6.369E+00  Zyy: 1.020E+01
Cw: 2.549E+03  x0: 0.000E+00  y0: 0.000E+00

MATERIAL PROPERTIES
Fyld: 5184.000  Fu: 8351.999

Actual Member Length: 10.000
Design Parameters
Kx: 1.00  Ky: 1.00  NSF: 0.85  SLF: 1.00  CSP: 1.00

COMPRESSION CLASSIFICATION (L/C: 1 LOC: 120.00)
Flange: NonSlender 7.22  N/A  15.89  Table.4.1a.Case1
Web: Slender 56.57  N/A  42.29  Table.4.1a.Case5

FLEXURE CLASSIFICATION (L/C: 1 LOC: 120.00)
Flange: Compact 7.22  10.79  28.38  Table.4.1b.Case10
Web: Compact 56.57  106.72  161.78  Table.4.1b.Case15

EXAMPLE PROBLEM NO. 1
-- PAGE NO. 27
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).

CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>10.25</td>
<td>421.2</td>
<td>0.024</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>10.25</td>
<td>421.2</td>
<td>0.024</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>0.000</td>
<td>374.0</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>0.000</td>
<td>256.6</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>0.000</td>
<td>316.0</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR SHEAR
**Application Examples**

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.000</td>
<td>113.7</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>10.63</td>
<td>156.5</td>
<td>0.068</td>
<td>Cl.G1</td>
<td>3</td>
</tr>
</tbody>
</table>

**CHECKS FOR BENDING**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>-182.5</td>
<td>257.6</td>
<td>0.708</td>
<td>Cl.F2.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>27.52</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt X</td>
<td>-182.5</td>
<td>220.9</td>
<td>0.826</td>
<td>Cl.F2.2</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECKS FOR AXIAL BEND INTERACTION**

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H1</td>
<td>0.827</td>
<td>Eq.H1-1b</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**EXAMPLE PROBLEM NO. 1**

---

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

**- Member : 7**

<table>
<thead>
<tr>
<th>Member No:</th>
<th>7</th>
<th>Profile: ST W21X44</th>
<th>(AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>PASS</td>
<td>Ratio:</td>
<td>0.707</td>
</tr>
<tr>
<td>Location:</td>
<td>10.00</td>
<td>Ref:</td>
<td>Eq.H1-1b</td>
</tr>
<tr>
<td>Pz:</td>
<td>10.25</td>
<td>T</td>
<td>Vy:</td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My:</td>
<td>0.000</td>
</tr>
</tbody>
</table>

**SLENDERNESS**

- Actual Slenderness Ratio : 95.097
- Allowable Slenderness Ratio : 300.000

**STRENGTH CHECKS**

| Critical L/C : 3 | Ratio : 0.707(PASS) | Condition : Eq.H1-1b |

**SECTION PROPERTIES (LOC: 10.00, PROPERTIES UNIT: IN )**

<table>
<thead>
<tr>
<th>Ag</th>
<th>1.300E+01</th>
<th>Axx</th>
<th>5.850E+00</th>
<th>Ayy</th>
<th>7.245E+00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ixx</td>
<td>8.430E+02</td>
<td>Iyy</td>
<td>2.070E+01</td>
<td>J</td>
<td>7.700E-01</td>
</tr>
<tr>
<td>Sxx+</td>
<td>8.145E+01</td>
<td>Sxx-</td>
<td>8.145E+01</td>
<td>Zxx</td>
<td>9.540E+01</td>
</tr>
<tr>
<td>Syy+</td>
<td>6.360E+00</td>
<td>Syy-</td>
<td>6.360E+00</td>
<td>Zyy</td>
<td>1.020E+01</td>
</tr>
<tr>
<td>Cw</td>
<td>2.540E+03</td>
<td>x0</td>
<td>0.000E+00</td>
<td>y0</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

**MATERIAL PROPERTIES**

| Fyld | 5184.000 | Fu | 8351.999 |

- Actual Member Length: 10.000
- Design Parameters (Rolled): Kx: 1.00 | Ky: 1.00 | NSF: 0.85 | SLF: 1.00 | CSP: 1.00

**COMPRESSION CLASSIFICATION (L/C: 1 LOC: 120.00)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>Flange: NonSlender</th>
<th>7.22</th>
<th>N/A</th>
<th>15.89</th>
<th>Table.4.1a.Case1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Web: Slender</td>
<td>56.57</td>
<td>N/A</td>
<td>42.29</td>
<td>Table.4.1a.Case5</td>
<td></td>
</tr>
</tbody>
</table>
FLEXURE CLASSIFICATION (L/C: 1 LOC: 120.00)

Flange: Compact 7.22 10.79 28.38 Table.4.1b.Case10
Web : Compact 56.57 106.72 161.78 Table.4.1b.Case15

EXAMPLE PROBLEM NO. 1

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>10.25</td>
<td>421.2</td>
<td>0.024</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>10.25</td>
<td>421.2</td>
<td>0.024</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>0.000</td>
<td>374.0</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>0.000</td>
<td>256.6</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>0.000</td>
<td>316.0</td>
<td>0.000</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.000</td>
<td>113.7</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>25.15</td>
<td>156.5</td>
<td>0.161</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>179.1</td>
<td>257.6</td>
<td>0.695</td>
<td>Cl.F2.1</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>27.52</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt X</td>
<td>179.1</td>
<td>257.6</td>
<td>0.695</td>
<td>Cl.F2.2</td>
<td>3</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H1</td>
<td>0.707</td>
<td>Eq.H1-1b</td>
<td>3</td>
</tr>
</tbody>
</table>
**SLENDERNESS**

Actual Slenderness Ratio : 47.018  
Allowable Slenderness Ratio : 200.000  
LOC : 0.00

**STRENGTH CHECKS**

Critical L/C : 1  
Ratio : 0.567 (PASS)  
Loc : 0.00  
Condition : Eq.H1-3a(H1-1b)

**SECTION PROPERTIES**  
(LOC: 0.00, PROPERTIES UNIT: IN)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>8.850E+00</td>
</tr>
<tr>
<td>Axx</td>
<td>5.182E+00</td>
</tr>
<tr>
<td>Ayy</td>
<td>3.726E+00</td>
</tr>
<tr>
<td>Ixx</td>
<td>2.910E+02</td>
</tr>
<tr>
<td>Iyy</td>
<td>1.960E+01</td>
</tr>
<tr>
<td>J</td>
<td>3.000E+00</td>
</tr>
<tr>
<td>Sxx+</td>
<td>4.217E+01</td>
</tr>
<tr>
<td>Sxx-</td>
<td>4.217E+01</td>
</tr>
<tr>
<td>Syy+</td>
<td>5.825E+00</td>
</tr>
<tr>
<td>Syy-</td>
<td>5.825E+00</td>
</tr>
<tr>
<td>Zxx</td>
<td>4.730E+00</td>
</tr>
<tr>
<td>Zyy</td>
<td>8.990E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>1.141E+03</td>
</tr>
<tr>
<td>x0</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>y0</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

**MATERIAL PROPERTIES**

Fyld: 5184.000  
Fu: 8351.999

**Actual Member Length:** 5.831

**Design Parameters**

Kx: 1.00  
Ky: 1.00  
NSF: 0.85  
SLF: 1.00  
CSP: 1.00

**COMPRESSION CLASSIFICATION**

(l/C: 3  LOC: 0.00)

<table>
<thead>
<tr>
<th>Case</th>
<th>Flange</th>
<th>Web</th>
</tr>
</thead>
<tbody>
<tr>
<td>NonSlender</td>
<td>8.74</td>
<td>N/A</td>
</tr>
<tr>
<td>Slender</td>
<td>48.26</td>
<td>N/A</td>
</tr>
</tbody>
</table>

**FLEXURE CLASSIFICATION**

(l/C: 3  LOC: 0.00)

<table>
<thead>
<tr>
<th>Case</th>
<th>Flange</th>
<th>Web</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compact</td>
<td>8.74</td>
<td>48.26</td>
</tr>
</tbody>
</table>

**EXAMPLE PROBLEM NO. 1**

**CHECKS FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

**CHECKS FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>36.32</td>
<td>274.6</td>
<td>0.132</td>
<td>Cl.E3</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>36.32</td>
<td>250.1</td>
<td>0.145</td>
<td>Cl.E3</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>36.32</td>
<td>261.2</td>
<td>0.139</td>
<td>Cl.E4</td>
<td>3</td>
</tr>
</tbody>
</table>

**CHECKS FOR SHEAR**

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
</table>
Shear X  0.000  100.7  0.000  Cl.G1  1  0.00  
Shear Y  16.86  80.48  0.210  Cl.G1  1  0.00

CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
</table>
| Flex. Yld. X  64.21  127.7  0.503  Cl.F2.1  1  0.00  
| Flex. Yld. Y  0.000  24.27  0.000  Cl.F6.1  1  0.00 |

CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
</table>
| Clause H1  0.574  Eq.H1-1b  1  0.00  
| Cl.H13-IP  0.567  Eq.H1-3a(H1-1b)  1  0.00  
| Cl.H13-OP  0.254  Eq.H1-3b  1  0.00 |

EXAMPLE PROBLEM NO. 1                                    -- PAGE NO. 32
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
*****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member : 9

Member No: 9  Profile: ST W14X30  (AISC SECTIONS)
Status: PASS  Ratio: 0.499  Loadcase: 1
Location: 5.83  Ref: Eq.H1-3a(H1-1b)
Pz: 35.72  C  Vy: 3.662  Vx: 0.000
Tz: 0.000  My: 0.000  Mx: -55.46

SLENDERNESS
Actual Slenderness Ratio : 47.018
Allowable Slenderness Ratio : 200.000  LOC : 0.00

STRENGTH CHECKS
Critical L/C : 1  Ratio : 0.499(PASS)
Loc : 5.83  Condition : Eq.H1-3a(H1-1b)

SECTION PROPERTIES (LOC: 5.83, PROPERTIES UNIT: IN )
Ag : 8.850E+00  Axx : 5.182E+00  Ayy : 3.726E+00
Ixx : 2.910E+02  Iyy : 1.960E+01  J : 3.800E-01
Syy+ : 5.825E+00  Syy- : 5.825E+00  Zyy : 8.990E+00
Cw : 1.141E+03  x0 : 0.000E+00  y0 : 0.000E+00

MATERIAL PROPERTIES
Fyld: 5184.000  Fu: 8351.999

Actual Member Length: 5.831
Design Parameters (Rolled)
Kx: 1.00  Ky: 1.00  NSF: 0.85  SLF: 1.00  CSP: 1.00

COMPRESSION CLASSIFICATION (L/C: 1 LOC: 0.00)
<table>
<thead>
<tr>
<th>l</th>
<th>1 p</th>
<th>1 r</th>
<th>CASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange: NonSlender  8.74  N/A  15.89  Table.4.1a.Case1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Web : Slender  48.26  N/A  42.29  Table.4.1a.Case5</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
EXAMPLE PROBLEM NO. 1                                    -- PAGE NO. 33

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
*****************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 9 Contd.

CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>35.81</td>
<td>274.6</td>
<td>0.130</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>35.81</td>
<td>250.1</td>
<td>0.143</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>35.81</td>
<td>261.2</td>
<td>0.137</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.000</td>
<td>100.7</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>9.060</td>
<td>80.48</td>
<td>0.113</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>-55.46</td>
<td>127.7</td>
<td>0.434</td>
<td>Cl.F2.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>24.27</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl.H1</td>
<td>0.506</td>
<td>Eq.H1-1b</td>
<td>1</td>
</tr>
<tr>
<td>Cl.H13-IP</td>
<td>0.499</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>1</td>
</tr>
<tr>
<td>Cl.H13-OP</td>
<td>0.319</td>
<td>Eq.H1-3b</td>
<td>1</td>
</tr>
</tbody>
</table>

EXAMPLE PROBLEM NO. 1                                    -- PAGE NO. 34

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
*****************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 10

Member No: 10 Profile: ST W14X30 (AISC SECTIONS)
Status: PASS Ratio: 0.637 Loadcase: 1
Location: 5.83 Ref: Eq.H1-3a(H1-1b)
Pz: 40.93 C Vy: -24.53 Vx: 0.000
### Application Examples

**EX. American Design Examples**

**EXAMPLE PROBLEM NO. 1**

#### SLENDERNESS

- **Actual Slenderness Ratio**: 47.018
- **Allowable Slenderness Ratio**: 200.000

#### STRENGTH CHECKS

- **Critical L/C**: 1
- **Ratio**: 0.637 (PASS)

#### SECTION PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>8.850E+00</td>
</tr>
<tr>
<td>Axx</td>
<td>5.182E+00</td>
</tr>
<tr>
<td>Ixx</td>
<td>2.910E+02</td>
</tr>
<tr>
<td>Sxx+</td>
<td>4.217E+01</td>
</tr>
<tr>
<td>Syy+</td>
<td>5.825E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>1.141E+03</td>
</tr>
</tbody>
</table>

#### MATERIAL PROPERTIES

- **Fyld**: 5184.000
- **Fu**: 8351.999

#### CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>286.7</td>
<td>Cl.D2</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>286.7</td>
<td>Cl.D2</td>
</tr>
</tbody>
</table>

#### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>41.02</td>
<td>274.6</td>
<td>Cl.E3</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>41.02</td>
<td>250.1</td>
<td>Cl.E3</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>41.02</td>
<td>261.2</td>
<td>Cl.E4</td>
</tr>
</tbody>
</table>

#### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>41.02</td>
<td>274.6</td>
<td>Cl.E3</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>41.02</td>
<td>250.1</td>
<td>Cl.E3</td>
</tr>
</tbody>
</table>

---

STAAD.Pro 4342 User Manual
<table>
<thead>
<tr>
<th>Member No:</th>
<th>11</th>
<th>Profile:</th>
<th>ST W14X30</th>
<th>Location:</th>
<th>5.83</th>
<th>Status:</th>
<th>PASS</th>
<th>Ratio:</th>
<th>0.637</th>
<th>Loadcase:</th>
<th>1</th>
<th>Reference:</th>
<th>Eq.H1-3a(H1-1b)</th>
<th>LOC:</th>
<th>5.83 ± 0.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz:</td>
<td>40.90</td>
<td>C</td>
<td>Vy:</td>
<td>-24.57</td>
<td>Vx:</td>
<td>0.000</td>
<td>Tz:</td>
<td>0.000</td>
<td>My:</td>
<td>0.000</td>
<td>Mx:</td>
<td>71.84</td>
<td>SLENDERNESS</td>
<td>Actual Slenderness Ratio: 47.018</td>
<td>Allowable Slenderness Ratio: 200.000</td>
</tr>
<tr>
<td>SECTIONS (LOC: 5.83, PROPERTIES UNIT: IN)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ag:</td>
<td>8.850E+00</td>
<td>Axx: 5.182E+00</td>
<td>Ayy: 3.726E+00</td>
<td>J: 3.800E-01</td>
<td>Sxx: 4.217E+01</td>
<td>Syy: 5.825E+00</td>
<td>Zyy: 8.990E+00</td>
<td>Cw: 1.141E+03</td>
<td>x0: 0.000E+00</td>
<td>y0: 0.000E+00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MATERIAL PROPERTIES</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fyd:</td>
<td>5184.000</td>
<td>Fu: 8351.999</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Actual Member Length: 5.831</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Design Parameters (Rolled)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Kx: 1.00</td>
<td>Ky: 1.00</td>
<td>NSF: 0.85</td>
<td>SLF: 1.00</td>
<td>CSP: 1.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>COMPRESSION CLASSIFICATION (L/C: 1 LOC: 0.00)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flange: NonSlender</td>
<td>8.74</td>
<td>N/A</td>
<td>15.89</td>
<td>Table.4.1a.Case1</td>
<td></td>
<td>Web: Slender</td>
<td>48.26</td>
<td>N/A</td>
<td>42.29</td>
<td>Table.4.1a.Case5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Example Problem No. 1

**STAAD.Pro Code Checking - AISC 360-16 LRFD (V1.1)**

**All units are - KIP FEET (unless otherwise noted).**

- **Member:** 11 Contd.

#### Checks for Axial Tension

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

#### Checks for Axial Compression

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>40.99</td>
<td>274.6</td>
<td>0.149</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>40.99</td>
<td>250.1</td>
<td>0.164</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>40.99</td>
<td>261.2</td>
<td>0.157</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

#### Checks for Shear

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.000</td>
<td>100.7</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>24.57</td>
<td>80.48</td>
<td>0.305</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

#### Checks for Bending

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>71.84</td>
<td>127.7</td>
<td>0.562</td>
<td>Cl.F2.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>24.27</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
</tbody>
</table>

#### Checks for Axial Bend Interaction

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl.H1</td>
<td>0.644</td>
<td>Eq.H1-1b</td>
<td>1</td>
</tr>
<tr>
<td>Cl.H13-IP</td>
<td>0.637</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>1</td>
</tr>
<tr>
<td>Cl.H13-OP</td>
<td>0.291</td>
<td>Eq.H1-3b</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**Example Problem No. 1**

**STAAD.Pro Code Checking - AISC 360-16 LRFD (V1.1)**

**All units are - KIP FEET (unless otherwise noted).**

- **Member:** 12

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST W14X30 (AISC Sections)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>PASS Ratio: 0.506 Loadcase: 1</td>
</tr>
<tr>
<td>Location:</td>
<td>5.83 Ref: Eq.H1-3a(H1-1b)</td>
</tr>
<tr>
<td>Pz:</td>
<td>38.62 C Vy: 2.651 Vx: 0.000</td>
</tr>
</tbody>
</table>

---

Application Examples

EX. American Design Examples

**STAAD.Pro 4344 User Manual**
SLENDERNESS
Actual Slenderness Ratio : 47.018
Allowable Slenderness Ratio : 200.000

STRENGTH CHECKS
Critical L/C : 1
Ratio : 0.506 (PASS)

SECTION PROPERTIES
Ag : 8.850E+00  Axx : 5.182E+00  Ayy : 3.726E+00
Ixx : 2.910E+02  Iyy : 1.960E+01  J   : 3.800E-01
Syy+ : 5.825E+00  Syy- : 5.825E+00  Zyy : 8.990E+00
Cw : 1.141E+03  x0 : 0.000E+00  y0 : 0.000E+00

MATERIAL PROPERTIES
Fyld: 5184.000  Fu: 8351.999

Actual Member Length: 5.831
Design Parameters
Kx: 1.00  Ky: 1.00  NSF: 0.85  SLF: 1.00  CSP: 1.00

Example Problem No. 1
STAAD.PRO Code Checking - AISC 360-16 LRFD (V1.1)

All units are - KIP  FEET (UNLESS OTHERWISE Noted).
Member : 12 Contd.

Checks for Axial Tension

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

Checks for Axial Compression

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>38.71</td>
<td>274.6</td>
<td>0.141</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>38.71</td>
<td>250.1</td>
<td>0.155</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>38.71</td>
<td>261.2</td>
<td>0.148</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

Checks for Shear

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
</table>
Shear X          0.000      100.7       0.000     Cl.G1           1    0.00
Shear Y          8.800      80.48       0.109     Cl.G1           3    0.00

CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>-55.68</td>
<td>127.7</td>
<td>0.436</td>
<td>Cl.F2.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>24.27</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H1</td>
<td>0.513</td>
<td>Eq.H1-1b</td>
<td>1</td>
</tr>
<tr>
<td>Cl.H13-IP</td>
<td>0.506</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>1</td>
</tr>
<tr>
<td>Cl.H13-OP</td>
<td>0.349</td>
<td>Eq.H1-3b</td>
<td>1</td>
</tr>
</tbody>
</table>

EXAMPLE PROBLEM NO. 1                                    -- PAGE NO. 40

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (VI.1)
****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     13

Member No:       13       Profile:  ST  W14X30              (AISC SECTIONS)
Status:        PASS       Ratio:         0.811       Loadcase:        3
Location:      0.00       Ref:      Eq.H1-3a(H1-1b)                        
Pz:       26.86     C     Vy:        19.06           Vx:      0.000        
Tz:       0.000           My:        0.000           Mx:      97.37        

SLENDERNESS
Actual Slenderness Ratio    :     47.018                                    
Allowable Slenderness Ratio :    200.000            LOC  :     0.00         

STRENGTH CHECKS
Critical L/C  :      3             Ratio     :        0.811(PASS)           
                             Loc  :    0.00            Condition :    Eq.H1-3a(H1-1b            

SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN )
Ag  :   8.850E+00     Axx :   5.182E+00     Ayy :   3.726E+00               
Ixx :   2.910E+02     Iyy :   1.960E+01     J   :   3.800E-01               
Sxx+:   4.217E+01     Sxx-:   4.217E+01     Zxx :   4.730E+01               
Syy+:   5.825E+00     Syy-:   5.825E+00     Zyy :   8.990E+00               
Cw  :   1.141E+03     x0  :   0.000E+00     y0  :   0.000E+00               

MATERIAL PROPERTIES
Fyld:        5184.000              Fu:        8351.999

Actual Member Length:         5.831

Design Parameters                                            (Rolled)
Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00

COMPRESSION CLASSIFICATION (L/C:     1 LOC:     0.00)

| Flange: NonSlender | 8.74 | N/A | 15.89 | Table.4.1a.Case1 |
| Web : Slender      | 48.26 | N/A | 42.29 | Table.4.1a.Case5 |
**FLEXURE CLASSIFICATION**

<table>
<thead>
<tr>
<th></th>
<th>L/C:</th>
<th>LOC:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange</td>
<td>1</td>
<td>0.00</td>
</tr>
<tr>
<td>Web</td>
<td>1p</td>
<td>1r</td>
</tr>
</tbody>
</table>

**CASE**

- **Flange**: Compact 8.74 10.79 28.38 Table 4.1b Case 10
- **Web**: Compact 48.26 106.72 161.78 Table 4.1b Case 15

---

**EXAMPLE PROBLEM NO. 1**

---

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

---

**CHECKS FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>0.000</td>
<td>286.7</td>
<td>0.000</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>39.00</td>
<td>274.6</td>
<td>0.142</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>39.00</td>
<td>250.1</td>
<td>0.156</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Tor. Buck</td>
<td>39.00</td>
<td>261.2</td>
<td>0.149</td>
<td>Cl.E4</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR SHEAR**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.000</td>
<td>100.7</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>19.06</td>
<td>80.48</td>
<td>0.237</td>
<td>Cl.G1</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR BENDING**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. X</td>
<td>97.37</td>
<td>127.7</td>
<td>0.762</td>
<td>Cl.F2.1</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Yld. Y</td>
<td>0.000</td>
<td>24.27</td>
<td>0.000</td>
<td>Cl.F6.1</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR AXIAL BEND INTERACTION**

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clue H1</td>
<td>0.816</td>
<td>Eq.H1-1b</td>
<td>3</td>
</tr>
<tr>
<td>Cl.H13-IP</td>
<td>0.811</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>3</td>
</tr>
<tr>
<td>Cl.H13-OP</td>
<td>0.345</td>
<td>Eq.H1-3b</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**EXAMPLE PROBLEM NO. 1**

---

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

---

**Member : 14**

<table>
<thead>
<tr>
<th>Member No.</th>
<th>14</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>PASS</td>
</tr>
<tr>
<td>Ratio:</td>
<td>0.187</td>
</tr>
<tr>
<td>Loadcase:</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**APPLICATION EXAMPLES**

- **EX. American Design Examples**
  - **STAAD.Pro 4347 User Manual**
SLENDERNESS
Actual Slenderness Ratio : 61.594
Allowable Slenderness Ratio : 200.000

STRENGTH CHECKS
Critical L/C : 3
Ratio : 0.187 (PASS)

SECTION PROPERTIES (LOC: 0.00, PROPERTIES UNIT: IN)
Ag : 1.000E+00
Axx : 4.700E-01
Ayy : 5.640E-01
Ixx : 9.095E-01
Iyy : 5.788E-01
J : 1.178E-02
Sxx+ : 5.634E-01
Sxx- : 5.604E-01
Zxx : 4.733E-01
Syy+ : 4.326E-01
Syy- : 5.093E-01
Zyy : 1.042E+00
Cw : 1.141E+03
x0 : -5.432E-01
y0 : -7.929E-01

MATERIAL PROPERTIES
Fyld: 5184.000
Fu: 8351.999

Actual Member Length: 3.905
Design Parameters
Kx: 1.00
Ky: 1.00
NSF: 0.85
SLF: 1.00
CSP: 1.00

COMPRESSION CLASSIFICATION (L/C: 3 LOC: 0.00)
Flange: Slender
13.30 N/A
Web : Slender
15.96 N/A

FLEXURE CLASSIFICATION (L/C: 3 LOC: 0.00)
Flange: Compact
13.30
Web : NonCompact
15.96

DEMAND CAPACITY RATIO REFERENCE
Ten. Yld. 2.432 32.40 0.075 Cl.D2 1 3.91
Ten. Rupt. 2.432 32.40 0.075 Cl.D2 1 3.91

DEMAND CAPACITY RATIO REFERENCE
Flex. Buck. X 4.778 27.15 0.176 Cl.E3 3 0.00
Flex. Buck. Y 4.778 25.55 0.187 Cl.E3 3 0.00
Flex. Buck. U 4.778 27.15 0.176 Cl.E3 3 0.00
Flex. Buck. V 4.778 25.55 0.187 Cl.E3 3 0.00

CHECKS FOR SHEAR
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.3608E-02</td>
<td>9.137</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.3342E-02</td>
<td>10.96</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

#### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>0.5782E-18</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-0.4985E-02</td>
<td>2.270</td>
<td>0.002</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>0.5782E-18</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-0.2891E-18</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-0.4985E-02</td>
<td>2.213</td>
<td>0.002</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
</tbody>
</table>

#### CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H2</td>
<td>0.187</td>
<td>Eq.H2-1</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**EXAMPLE PROBLEM NO. 1**

-- PAGE NO. 44

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

*ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).*

- **Member:** 15

---

**Member No:** 15  | **Profile:** ST L30253  | (AISC SECTIONS)
**Status:** PASS  | **Ratio:** 0.127  | **Loadcase:** 3
**Location:** 3.79  | **Ref:** Eq.H2-1

<table>
<thead>
<tr>
<th>Pz</th>
<th>4.010</th>
<th>T</th>
<th>Vy</th>
<th>0.2843E-03</th>
<th>Vx</th>
<th>0.2206E-03</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tz</td>
<td>0.000</td>
<td></td>
<td>My</td>
<td>0.4628E-02</td>
<td>Mx</td>
<td>-0.4286E-02</td>
</tr>
</tbody>
</table>

**SLENDERNESS**

- **Actual Slenderness Ratio:** 102.522
- **Allowable Slenderness Ratio:** 300.000  | **LOC:** 0.00

**STRENGTH CHECKS**

- **Critical L/C:** 3  | **Ratio:** 0.127 (PASS)
- **Loc:** 3.79  | **Condition:** Eq.H2-1

**SECTION PROPERTIES** (LOC: 3.79, PROPERTIES UNIT: IN )

<table>
<thead>
<tr>
<th>Ag</th>
<th>1.000E+00</th>
<th>Axx</th>
<th>4.708E-01</th>
<th>Ayy</th>
<th>5.640E-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ixx</td>
<td>9.095E-01</td>
<td>Iyy</td>
<td>5.788E-01</td>
<td>J</td>
<td>1.178E-02</td>
</tr>
<tr>
<td>Sxx+</td>
<td>5.634E-01</td>
<td>Sxx-</td>
<td>5.604E-01</td>
<td>Zxx</td>
<td>4.733E-01</td>
</tr>
<tr>
<td>Syy+</td>
<td>4.362E-01</td>
<td>Syy-</td>
<td>5.093E-01</td>
<td>Zyy</td>
<td>1.042E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>1.141E+03</td>
<td>x0</td>
<td>-5.432E-01</td>
<td>y0</td>
<td>-7.929E-01</td>
</tr>
</tbody>
</table>

**MATERIAL PROPERTIES**

<table>
<thead>
<tr>
<th>Fyld</th>
<th>5184.000</th>
<th>Fu</th>
<th>8351.999</th>
</tr>
</thead>
</table>

- **Actual Member Length:** 6.500

**Design Parameters**

(rolled)

<table>
<thead>
<tr>
<th>Kx</th>
<th>1.00</th>
<th>Ky</th>
<th>1.00</th>
<th>NSF</th>
<th>0.85</th>
<th>SLF</th>
<th>1.00</th>
<th>CSP</th>
<th>1.00</th>
</tr>
</thead>
</table>

**COMPRESSION CLASSIFICATION (L/C: 1 LOC: 0.00)**

<table>
<thead>
<tr>
<th>Flange</th>
<th>Slender</th>
<th>13.30</th>
<th>N/A</th>
<th>12.77</th>
<th>Table.4.1a.Case3</th>
</tr>
</thead>
</table>
EX. American Design Examples

STAAD.Pro 4350 User Manual

Application Examples

Web : Slender 15.96 N/A 12.77 Table.4.1a.Case3

FLEXURE CLASSIFICATION (L/C: 1 LOC: 0.00)

Flange: Compact 13.30 15.33 25.83 Table.4.1b.Case12 Web : NonCompact 15.96 15.33 25.83 Table.4.1b.Case12

EXAMPLE PROBLEM NO. 1 -- PAGE NO. 45

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 15 Contd.

CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>4.017</td>
<td>32.40</td>
<td>0.124</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>4.017</td>
<td>32.40</td>
<td>0.124</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>1.983</td>
<td>22.47</td>
<td>0.088</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>1.983</td>
<td>18.63</td>
<td>0.106</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
<td>1.983</td>
<td>22.47</td>
<td>0.088</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
<td>1.983</td>
<td>18.63</td>
<td>0.106</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.3287E-02</td>
<td>9.137</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.3045E-02</td>
<td>10.96</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>-5782E-18</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-8018E-02</td>
<td>2.270</td>
<td>0.004</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>-5782E-18</td>
<td>1.509</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-5782E-18</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-8018E-02</td>
<td>2.213</td>
<td>0.004</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H2</td>
<td>0.127</td>
<td>Eq.H2-1</td>
<td>3</td>
</tr>
</tbody>
</table>

EXAMPLE PROBLEM NO. 1 -- PAGE NO. 46

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 16
Member No: 16  Profile: ST L30253  (AISC SECTIONS)
Status: PASS  Ratio: 0.755  Loadcase: 1
Location: 3.91  Ref: Eq.H2-1
Pz: 24.23  T  Vy: 0.2693E-03  Vx: 0.2907E-03
Tz: 0.000  My: 0.1203E-01  Mx: -.1114E-01

SLENDERNESS
Actual Slenderness Ratio : 123.188
Allowable Slenderness Ratio : 300.000  LOC : 0.00

STRENGTH CHECKS
Critical L/C : 1  Ratio : 0.755(PASS)
Loc : 3.91  Condition : Eq.H2-1

SECTION PROPERTIES (LOC: 3.91, PROPERTIES UNIT: IN )
Ag : 1.000E+00  Axx : 4.700E-01  Ayy : 5.640E-01
Ixx : 9.095E-01  Iyy : 5.788E-01  J : 1.178E-02
Sxx+: 5.634E-01  Sxx-: 5.604E-01  Zxx : 4.733E-01
Syy+: 4.326E-01  Syy-: 5.093E-01  Zyy : 1.042E+00
Cw : 1.141E+03  x0 : -5.432E-01  y0 : -7.929E-01

MATERIAL PROPERTIES
Fyld: 5184.000  Fu: 8351.999

Actual Member Length: 7.810
Design Parameters (Rolled)
Kx: 1.00  Ky: 1.00  NSF: 0.85  SLF: 1.00  CSP: 1.00

CHECKS FOR AXIAL TENSION
DEMAND  CAPACITY  RATIO  REFERENCE  L/C  LOC
Ten. Yld.  24.24  32.40  0.748  Cl.D2  1  7.81
Ten. Rupt.  24.24  32.40  0.748  Cl.D2  1  7.81

CHECKS FOR AXIAL COMPRESSION
DEMAND  CAPACITY  RATIO  REFERENCE  L/C  LOC
Flex. Buck. X 0.000  19.49  0.000  Cl.E3  1  0.00
Flex. Buck. Y 0.000  14.57  0.000  Cl.E3  1  0.00
Flex. Buck. U 0.000  19.49  0.000  Cl.E3  1  0.00

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 16 Contd.
### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.6519E-02</td>
<td>9.137</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.6039E-02</td>
<td>10.96</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>-.1156E-17</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-.1639E-01</td>
<td>2.270</td>
<td>0.007</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>-.1156E-17</td>
<td>1.445</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-.1156E-17</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-.1639E-01</td>
<td>2.213</td>
<td>0.007</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Crieteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H2</td>
<td>0.755</td>
<td>Eq.H2-1</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**EXAMPLE PROBLEM NO. 1**

---

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

---

**STAAD.Pro**

---

**4352 User Manual**
**Application Examples**

**EX. American Design Examples**

---

**COMPRESSION CLASSIFICATION (L/C: 93.72)**

| Flange: Slender | 13.30 | N/A | 12.77 | Table 4.1a.Case3 |
| Web : Slender  | 15.96 | N/A | 12.77 | Table 4.1a.Case3 |

**FLEXURE CLASSIFICATION (L/C: 93.72)**

| Flange: Compact | 13.30 | 15.33 | 25.83 | Table 4.1b.Case12 |
| Web : NonCompact | 15.96 | 15.33 | 25.83 | Table 4.1b.Case12 |

---

**CHECKS FOR AXIAL TENSION**

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>23.48</td>
<td>32.40</td>
<td>0.725</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>23.48</td>
<td>32.40</td>
<td>0.725</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR AXIAL COMPRESSION**

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>0.000</td>
<td>19.49</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>0.000</td>
<td>14.57</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
<td>0.000</td>
<td>19.49</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
<td>0.000</td>
<td>14.57</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR SHEAR**

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.6532E-02</td>
<td>9.137</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.6050E-02</td>
<td>10.96</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR BENDING**

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>-0.1156E-17</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-0.1638E-01</td>
<td>2.270</td>
<td>0.007</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>-0.1156E-17</td>
<td>1.445</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-0.1156E-17</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-0.1638E-01</td>
<td>2.213</td>
<td>0.007</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
</tbody>
</table>

---

**CHECKS FOR AXIAL BEND INTERACTION**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H2</td>
<td>0.730</td>
<td>Eq.H2-1</td>
<td>3</td>
</tr>
</tbody>
</table>

---

**Example Problem No. 1**

STAAD.Pro Code Checking - AISC 360-16 LRFD (V1.1)

---

**All units are - KIP FEET (UNLESS OTHERWISE Noted).**

- Member : 17 Contd.
### SLENDERNESS

- **Actual Slenderness Ratio:** 102.522
- **Allowable Slenderness Ratio:** 200.000
- **Location:** 0.00

### STRENGTH CHECKS

- **Critical L/C:** 3
- **Ratio:** 0.279 (PASS)
- **Condition:** Cl.E3

### SECTION PROPERTIES (LOC: 0.00, PROPERTIES UNIT: IN)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>1.000E+00</td>
</tr>
<tr>
<td>Axx</td>
<td>4.700E-01</td>
</tr>
<tr>
<td>Ayy</td>
<td>5.640E-01</td>
</tr>
<tr>
<td>Ixx</td>
<td>9.095E-01</td>
</tr>
<tr>
<td>Iyy</td>
<td>5.788E-01</td>
</tr>
<tr>
<td>J</td>
<td>1.178E-02</td>
</tr>
<tr>
<td>Sxx+</td>
<td>5.634E-01</td>
</tr>
<tr>
<td>Sxx-</td>
<td>5.604E-01</td>
</tr>
<tr>
<td>Zxx</td>
<td>4.733E-01</td>
</tr>
<tr>
<td>Syy+</td>
<td>4.326E-01</td>
</tr>
<tr>
<td>Syy-</td>
<td>5.093E-01</td>
</tr>
<tr>
<td>Zyy</td>
<td>1.042E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>1.141E+03</td>
</tr>
<tr>
<td>x0</td>
<td>-5.432E-01</td>
</tr>
<tr>
<td>y0</td>
<td>-7.929E-01</td>
</tr>
</tbody>
</table>

### MATERIAL PROPERTIES

- **Fyld:** 5184.000
- **Fu:** 8351.999

### Design Parameters

- **Kx:** 1.00
- **Ky:** 1.00
- **NSF:** 0.85
- **SLF:** 1.00
- **CSP:** 1.00

### COMPRESSION CLASSIFICATION (L/C: 3 LOC: 0.00)

| Flange | Slender | 13.30 | N/A | 12.77 | Table.4.1a.Case3 |
| Web    | Slender | 15.96 | N/A | 12.77 | Table.4.1a.Case3 |

### FLEXURE CLASSIFICATION (L/C: 3 LOC: 0.00)

| Flange | Compact | 13.30 | 15.33 | 25.83 | Table.4.1b.Case12 |
| Web    | NonCompact | 15.96 | 15.33 | 25.83 | Table.4.1b.Case12 |

---

**EXAMPLE PROBLEM NO. 1**

### CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>1.124</td>
<td>32.40</td>
<td>0.035</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>1.124</td>
<td>32.40</td>
<td>0.035</td>
<td>Cl.D2</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>5.199</td>
<td>22.47</td>
<td>0.231</td>
<td>Cl.E3</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>5.199</td>
<td>18.63</td>
<td>0.279</td>
<td>Cl.E3</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
<td>5.199</td>
<td>22.47</td>
<td>0.231</td>
<td>Cl.E3</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
<td>5.199</td>
<td>18.63</td>
<td>0.279</td>
<td>Cl.E3</td>
<td>3</td>
</tr>
</tbody>
</table>

### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.3299E-02</td>
<td>9.137</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.3055E-02</td>
<td>10.96</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>0.5782E-18</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-0.8158E-02</td>
<td>2.270</td>
<td>0.004</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>-0.5782E-18</td>
<td>1.509</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-0.5782E-18</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-0.8158E-02</td>
<td>2.213</td>
<td>0.004</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H2</td>
<td>0.279</td>
<td>Eq.H2-1</td>
<td>3</td>
</tr>
</tbody>
</table>

Example Problem No. 1

---

STAAD.Pro Code Checking - AISC 360-16 LRFD (V1.1)

---

All units are - KIP FEET (UNLESS OTHERWISE Noted).

---

STAAD.Pro 4355 User Manual
### Actual Member Length:
3.905

### Design Parameters (Rolled)
- Kx: 1.00
- Ky: 1.00
- NSF: 0.85
- SLF: 1.00
- CSP: 1.00

### COMPRESSION CLASSIFICATION (L/C: 1 LOC: 0.00)
- Flange: Slender
  - 13.30 (N/A)
  - 12.77
  - Table.4.1a.Case3
- Web: Slender
  - 15.96 (N/A)
  - 12.77
  - Table.4.1a.Case3

### FLEXURE CLASSIFICATION (L/C: 1 LOC: 0.00)
- Flange: Compact
  - 13.30
  - 15.33
  - 25.83
  - Table.4.1b.Case12
- Web: NonCompact
  - 15.96
  - 15.33
  - 25.83
  - Table.4.1b.Case12

---

### EXAMPLE PROBLEM NO. 1

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

All units are - KIP FEET (UNLESS OTHERWISE Noted).

- Member: 19 Contd.

### CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>6.284</td>
<td>32.40</td>
<td>0.194</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>6.284</td>
<td>32.40</td>
<td>0.194</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>1.287</td>
<td>27.15</td>
<td>0.047</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>1.287</td>
<td>25.55</td>
<td>0.050</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
<td>1.287</td>
<td>27.15</td>
<td>0.047</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
<td>1.287</td>
<td>25.55</td>
<td>0.050</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.3574E-02</td>
<td>9.137</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.3310E-02</td>
<td>10.96</td>
<td>0.000</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>0.2891E-18</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-0.4989E-02</td>
<td>2.270</td>
<td>0.002</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>-0.2891E-18</td>
<td>1.655</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-0.2891E-18</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-0.4989E-02</td>
<td>2.213</td>
<td>0.002</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
EXAMPLE PROBLEM NO. 1  -- PAGE NO.  54
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
*****************************************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).
- Member :    20

Member No:    20  Profile: ST L30253   (AISC SECTIONS)
Status:        PASS       Ratio:         0.571       Loadcase:        3
Location:      3.75       Ref:      Eq.H2-1                                
Pz:       18.23     T     Vy:       -.4719E-04       Vx:     -.5095E-04    
Tz:       0.000           My:       0.1347E-01       Mx:     -.1248E-01    

SLENDERNESS
Actual Slenderness Ratio    :    118.295                                    
Allowable Slenderness Ratio :    300.000            LOC  :     0.00         

STRENGTH CHECKS
Critical L/C  :      3             Ratio     :        0.571(PASS)           
Loc : 3.75            Condition :    Eq.H2-1                   

SECTION PROPERTIES  (LOC:     3.75, PROPERTIES UNIT: IN  )                  
Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01               
Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02               
Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01               
Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00               
Cw  :   1.141E+03     x0  :  -5.432E-01     y0  :  -7.929E-01               

MATERIAL PROPERTIES
Fyld:        5184.000              Fu:        8351.999                      

Actual Member Length:         7.500                                         
Design Parameters                                            (Rolled)       
Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00          

COMPRESS TION CLASSIFICATION (L/C:      1 LOC:    90.00)                      
    l           l p         l r       CASE               
Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3     
Web : Slender         15.96       N/A      12.77     Table.4.1a.Case3     

FLEXURE CLASSIFICATION     (L/C:      1 LOC:    90.00)                      
    l           l p         l r       CASE               
Flange: Compact        13.30      15.33     25.83     Table.4.1b.Case12     
Web : Noncompact      15.96      15.33     25.83     Table.4.1b.Case12     

EXAMPLE PROBLEM NO. 1  -- PAGE NO.  55
STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)
*****************************************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).
- Member :    20 Contd.

CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Demanding</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>18.23</td>
<td>32.40</td>
<td>0.563</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>18.23</td>
<td>32.40</td>
<td>0.563</td>
<td>Cl.D2</td>
<td>3</td>
</tr>
</tbody>
</table>
### Checks for Axial Compression

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>Loc</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>0.000</td>
<td>20.27</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>0.000</td>
<td>15.51</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
<td>0.000</td>
<td>20.27</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
<td>0.000</td>
<td>15.51</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
</tr>
</tbody>
</table>

### Checks for Shear

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>Loc</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.9564E-02</td>
<td>9.137</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.8858E-02</td>
<td>10.96</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
</tr>
</tbody>
</table>

### Checks for Bending

<table>
<thead>
<tr>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
<th>L/C</th>
<th>Loc</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>0.1156E-17</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>3</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-.2312E-01</td>
<td>2.270</td>
<td>0.010</td>
<td>Cl.F10.1</td>
<td>1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>-.1156E-17</td>
<td>1.460</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>3</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-.1156E-17</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>3</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-.2312E-01</td>
<td>2.213</td>
<td>0.010</td>
<td>Cl.F10.3</td>
<td>1</td>
</tr>
</tbody>
</table>

### Checks for Axial Bend Interaction

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Criteria</th>
<th>L/C</th>
<th>Loc</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.571</td>
<td>Eq.H2-1</td>
<td>3</td>
<td>3.75</td>
</tr>
</tbody>
</table>

---

**EXAMPLE PROBLEM NO. 1**

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

- Member: 21

**Member No:** 21  **Profile:** ST L30253

**Status:** PASS  **Ratio:** 0.564  **Loadcase:** 1

**Location:** 3.75  **Ref:** Eq.H2-1

**Pz:** 17.90  **Tz:** 0.000  **Vx:** -.2076E-03  **My:** 0.1788E-01

---

**SLENDERNESS**

**Actual Slenderness Ratio:** 118.295

**Allowable Slenderness Ratio:** 300.000  **LOC:** 0.00

---

**STRENGTH CHECKS**

**Critical L/C:** 1  **Ratio:** 0.564(PASS)  **Condition:** Eq.H2-1

**Loc:** 3.75

---

**SECTION PROPERTIES**

<table>
<thead>
<tr>
<th>Ag</th>
<th>Axx</th>
<th>Ayy</th>
<th>J</th>
<th>Sxx+</th>
<th>Sxx-</th>
<th>Syy+</th>
<th>Syy-</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.008E+00</td>
<td>4.708E-01</td>
<td>5.640E-01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9.995E-01</td>
<td>5.788E-01</td>
<td>1.178E-02</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.634E-01</td>
<td>5.604E-01</td>
<td>4.733E-01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.326E-01</td>
<td>5.093E-01</td>
<td>1.042E+00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

EX. American Design Examples

---

#### MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld:</td>
<td>5184.000</td>
</tr>
<tr>
<td>Fu:</td>
<td>8351.999</td>
</tr>
</tbody>
</table>

#### Actual Member Length

- Value: 7.500

#### Design Parameters

- Kx: 1.00
- Ky: 1.00
- NSF: 0.85
- SLF: 1.00
- CSP: 1.00

#### COMPRESSION CLASSIFICATION

<table>
<thead>
<tr>
<th>Type</th>
<th>L/C</th>
<th>LOC</th>
<th>Flange</th>
<th>Web</th>
<th>Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slender</td>
<td>1</td>
<td>90</td>
<td>13.30</td>
<td>15.96</td>
<td>Table.4.1a.Case3</td>
</tr>
<tr>
<td>Compact</td>
<td>1</td>
<td>90</td>
<td>13.30</td>
<td>15.96</td>
<td>Table.4.1b.Case12</td>
</tr>
</tbody>
</table>

#### FLEXURE CLASSIFICATION

<table>
<thead>
<tr>
<th>Type</th>
<th>L/C</th>
<th>LOC</th>
<th>Flange</th>
<th>Web</th>
<th>Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compact</td>
<td>1</td>
<td>90</td>
<td>13.30</td>
<td>15.96</td>
<td>Table.4.1b.Case12</td>
</tr>
<tr>
<td>NonCompact</td>
<td>1</td>
<td>90</td>
<td>15.33</td>
<td>25.83</td>
<td>Table.4.1b.Case12</td>
</tr>
</tbody>
</table>

---

#### EXAMPLE PROBLEM NO. 1

- Member: 21 Contd.

#### CHECKS FOR AXIAL TENSION

<table>
<thead>
<tr>
<th>Type</th>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>17.90</td>
<td>32.40</td>
<td>0.553</td>
<td>Cl.D2</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>17.90</td>
<td>32.40</td>
<td>0.553</td>
<td>Cl.D2</td>
</tr>
</tbody>
</table>

#### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th>Type</th>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>0.000</td>
<td>20.27</td>
<td>0.000</td>
<td>Cl.E3</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>0.000</td>
<td>15.51</td>
<td>0.000</td>
<td>Cl.E3</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
<td>0.000</td>
<td>20.27</td>
<td>0.000</td>
<td>Cl.E3</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
<td>0.000</td>
<td>15.51</td>
<td>0.000</td>
<td>Cl.E3</td>
</tr>
</tbody>
</table>

#### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th>Type</th>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.9551E-02</td>
<td>9.137</td>
<td>0.001</td>
<td>Cl.G1</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.8846E-02</td>
<td>10.96</td>
<td>0.001</td>
<td>Cl.G1</td>
</tr>
</tbody>
</table>

#### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th>Type</th>
<th>Demand</th>
<th>Capacity</th>
<th>Ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>0.2313E-17</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-2.2438E-01</td>
<td>2.270</td>
<td>0.011</td>
<td>Cl.F10.1</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>0.2313E-17</td>
<td>1.526</td>
<td>0.000</td>
<td>Cl.F10.2</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-1.156E-17</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-2.2438E-01</td>
<td>2.213</td>
<td>0.011</td>
<td>Cl.F10.3</td>
</tr>
</tbody>
</table>
CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th>Clause H2</th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.564</td>
<td>Eq.H2-1</td>
<td>1</td>
<td>3.75</td>
</tr>
</tbody>
</table>

EXAMPLE PROBLEM NO. 1

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 22

<table>
<thead>
<tr>
<th>Member No:</th>
<th>22</th>
<th>Profile:  ST L30253              (AISC SECTIONS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status:</td>
<td>PASS</td>
<td>Ratio:   0.620       Loadcase:        1</td>
</tr>
<tr>
<td>Location:</td>
<td>3.75</td>
<td>Ref:      Eq.H2-1</td>
</tr>
<tr>
<td>Pz:</td>
<td>19.72</td>
<td>T</td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My:      0.1798E-01       Mx:     -.1665E-01</td>
</tr>
</tbody>
</table>

STRENGTH CHECKS

Critical L/C : 1       Ratio : 0.620(PASS)       Loadcase: 1

SECTION PROPERTIES (LOC: 3.75, PROPERTIES UNIT: IN)

Ag : 1.000E+00     Axx : 4.700E-01     Ayy : 5.640E-01
Ixx : 9.095E-01     Iyy : 5.788E-01     J : 1.178E-02
Sxx+: 5.634E-01     Sxx-: 5.604E-01     Zxx : 4.733E-01
Syy+: 4.326E-01     Syy-: 5.093E-01     Zyy : 1.042E+00
Cw : 1.141E+03     x0  : -5.432E-01     y0  : -7.929E-01

MATERIAL PROPERTIES

Fyld: 5184.000       Fu: 8351.999

Actual Member Length: 7.500

Design Parameters (Rolled)

Kx: 1.00  Ky: 1.00  NSF: 0.85  SLF: 1.00  CSP: 1.00

COMPRESSION CLASSIFICATION (L/C: 1 LOC: 90.00)

<table>
<thead>
<tr>
<th>Flange</th>
<th>Slender</th>
<th>13.30</th>
<th>N/A</th>
<th>12.77</th>
<th>Table.4.1a.Case3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Web</td>
<td>Slender</td>
<td>15.96</td>
<td>N/A</td>
<td>12.77</td>
<td>Table.4.1a.Case3</td>
</tr>
</tbody>
</table>

FLEXURE CLASSIFICATION (L/C: 1 LOC: 90.00)

<table>
<thead>
<tr>
<th>Flange</th>
<th>Compact</th>
<th>13.30</th>
<th>15.33</th>
<th>25.83</th>
<th>Table.4.1b.Case12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Web</td>
<td>NonCompact</td>
<td>15.96</td>
<td>15.33</td>
<td>25.83</td>
<td>Table.4.1b.Case12</td>
</tr>
</tbody>
</table>

CHECKS FOR AXIAL TENSION

---

EXAMPLE PROBLEM NO. 2

---

EXAMPLE PROBLEM NO. 1

---

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 22 Contd.
### Application Examples
EX. American Design Examples

<table>
<thead>
<tr>
<th></th>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ten. Yld.</td>
<td>19.72</td>
<td>32.40</td>
<td>0.609</td>
<td>Cl.D2</td>
<td>1</td>
<td>0.00</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
<td>19.72</td>
<td>32.40</td>
<td>0.609</td>
<td>Cl.D2</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL COMPRESSION

<table>
<thead>
<tr>
<th></th>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Buck. X</td>
<td>0.000</td>
<td>20.27</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
<td>0.00</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
<td>0.000</td>
<td>15.51</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
<td>0.00</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
<td>0.000</td>
<td>20.27</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
<td>0.00</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
<td>0.000</td>
<td>15.51</td>
<td>0.000</td>
<td>Cl.E3</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECKS FOR SHEAR

<table>
<thead>
<tr>
<th></th>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear X</td>
<td>0.9515E-02</td>
<td>9.137</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
<td>0.00</td>
</tr>
<tr>
<td>Shear Y</td>
<td>0.8813E-02</td>
<td>10.96</td>
<td>0.001</td>
<td>Cl.G1</td>
<td>1</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### CHECKS FOR BENDING

<table>
<thead>
<tr>
<th></th>
<th>DEMAND</th>
<th>CAPACITY</th>
<th>RATIO</th>
<th>REFERENCE</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex. Yld. U</td>
<td>-.2313E-17</td>
<td>1.752</td>
<td>0.000</td>
<td>Cl.F10.1</td>
<td>1</td>
<td>2.50</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
<td>-.2451E-01</td>
<td>2.270</td>
<td>0.011</td>
<td>Cl.F10.1</td>
<td>1</td>
<td>3.75</td>
</tr>
<tr>
<td>L-T-B Abt U</td>
<td>-.2313E-17</td>
<td>1.617</td>
<td>0.000</td>
<td>Cl.F10.2</td>
<td>1</td>
<td>2.50</td>
</tr>
<tr>
<td>F-L-B Abt U</td>
<td>-.2313E-17</td>
<td>1.709</td>
<td>0.000</td>
<td>Cl.F10.3</td>
<td>1</td>
<td>2.50</td>
</tr>
<tr>
<td>F-L-B Abt V</td>
<td>-.2451E-01</td>
<td>2.213</td>
<td>0.011</td>
<td>Cl.F10.3</td>
<td>1</td>
<td>3.75</td>
</tr>
</tbody>
</table>

### CHECKS FOR AXIAL BEND INTERACTION

<table>
<thead>
<tr>
<th></th>
<th>RATIO</th>
<th>CRITERIA</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clause H2</td>
<td>0.620</td>
<td>Eq.H2-1</td>
<td>1</td>
<td>3.75</td>
</tr>
</tbody>
</table>

---

**EXAMPLE PROBLEM NO. 1**

---

**STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)**

---

**ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).**

---

**STAAD.Pro 4361 User Manual**
### Application Examples

EX. American Design Examples

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ag</td>
<td>1.000E+00</td>
</tr>
<tr>
<td>Axx</td>
<td>4.700E-01</td>
</tr>
<tr>
<td>Ayy</td>
<td>5.640E-01</td>
</tr>
<tr>
<td>Ixx</td>
<td>9.095E-01</td>
</tr>
<tr>
<td>Iyy</td>
<td>5.788E-01</td>
</tr>
<tr>
<td>J</td>
<td>1.178E-02</td>
</tr>
<tr>
<td>Sxx+</td>
<td>4.326E-01</td>
</tr>
<tr>
<td>Sxx-</td>
<td>4.504E-01</td>
</tr>
<tr>
<td>Zxx</td>
<td>4.733E-01</td>
</tr>
<tr>
<td>Syy+</td>
<td>5.093E-01</td>
</tr>
<tr>
<td>Syy-</td>
<td>5.604E-01</td>
</tr>
<tr>
<td>Zyy</td>
<td>1.042E+00</td>
</tr>
<tr>
<td>Cw</td>
<td>1.141E+03</td>
</tr>
<tr>
<td>x0</td>
<td>-5.432E-01</td>
</tr>
<tr>
<td>y0</td>
<td>-7.929E-01</td>
</tr>
</tbody>
</table>

#### MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fyld</td>
<td>5184.000</td>
</tr>
<tr>
<td>Fu</td>
<td>8351.999</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Actual Member Length</th>
<th>7.500</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Design Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kx:</td>
<td>1.00</td>
</tr>
<tr>
<td>Ky:</td>
<td>1.00</td>
</tr>
<tr>
<td>NSF:</td>
<td>0.85</td>
</tr>
<tr>
<td>SLF:</td>
<td>1.00</td>
</tr>
<tr>
<td>CSP:</td>
<td>1.00</td>
</tr>
</tbody>
</table>

#### COMPRESSION CLASSIFICATION

<table>
<thead>
<tr>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl.D2</td>
<td>1</td>
<td>0.00</td>
</tr>
<tr>
<td>Table.4.1a.Case3</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### FLEXURE CLASSIFICATION

<table>
<thead>
<tr>
<th>Reference</th>
<th>L/C</th>
<th>LOC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl.E3</td>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>Table.4.1b.Case12</td>
<td>3</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### EXAMPLE PROBLEM NO. 1

---

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 23 Contd.

<table>
<thead>
<tr>
<th>CHECKS FOR AXIAL TENSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Demand</td>
</tr>
<tr>
<td>-------</td>
</tr>
<tr>
<td>Ten. Yld.</td>
</tr>
<tr>
<td>Ten. Rupt.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CHECKS FOR AXIAL COMPRESSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Demand</td>
</tr>
<tr>
<td>-------</td>
</tr>
<tr>
<td>Flex. Buck. X</td>
</tr>
<tr>
<td>Flex. Buck. Y</td>
</tr>
<tr>
<td>Flex. Buck. U</td>
</tr>
<tr>
<td>Flex. Buck. V</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CHECKS FOR SHEAR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Demand</td>
</tr>
<tr>
<td>-------</td>
</tr>
<tr>
<td>Shear X</td>
</tr>
<tr>
<td>Shear Y</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CHECKS FOR BENDING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Demand</td>
</tr>
<tr>
<td>-------</td>
</tr>
<tr>
<td>Flex. Yld. U</td>
</tr>
<tr>
<td>Flex. Yld. V</td>
</tr>
</tbody>
</table>
EX. US-2 Area Load Generation on Floor Structure

A floor structure (bound by global X-Z axis) made up of steel beams is subjected to area load (i.e., load/area of floor). Load generation based on one-way distribution is illustrated in this example.

In the case of loads such as joint loads and member loads, the magnitude and direction of the load at the applicable joints and members is directly known from the input. However, the area load is a different sort of load where a load intensity on the given area has to be converted to joint and member loads. The calculations required to perform this conversion are done only during the analysis. Consequently, the loads generated from the AREA LOAD command can be viewed only after the analysis is completed.

This problem is installed with the program by default to C:sers\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-2 Area Load Generation on Floor Structure.STD when you install the program.
Figure 455: Example Problem No. 2

Where:

W1 = 5 ft, W2 = 3.5 ft, W3 = 5.5 ft, L1 = 7 ft, L2 = 15 ft, L3 = 10 ft

Actual input is shown in bold lettering followed by explanation.

STAAD FLOOR A FLOOR FRAME DESIGN WITH AREA LOAD

Every input has to start with the term STAAD. The term FLOOR signifies that the structure is a floor structure and the structure is in the x – z plane.

UNIT FT KIP

Defines the input units for the data that follows.

JOINT COORDINATES
1 0. 0. 0. 5 20. 0. 0. ; 7 5. 0. 10.
8 10. 0. 10. ; 9 13. 0. 10. ; 10 15. 0. 10. ; 11 16.5 0. 10.
12 20. 0. 10. ; 13 0. 0. 25. ; 14 5. 0. 25. ; 15 11. 0. 25.
16 16.5 0. 25 ; 17 20. 0. 25. 18 0. 0. 28.
19 20. 0. 28. ; 20 0. 0. 35. ; 21 20. 0. 35.
Joint numbers followed by X, Y and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). Joints between 1 and 5 (i.e., 2, 3, 4) are generated in the first line of input, taking advantage of the equal spacing between the joints (see TR.11 Joint Coordinates Specification on page 2425 for more information).

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
19 7 14 ; 20 2 7 ; 21 9 15
22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
26 17 19 ; 27 12 17 ; 28 5 12
```

Defines the members by the joints to which they are connected.

```
MEMB PROP AMERICAN
1 TO 28 TABLE ST W12X26
```

Member properties are specified from the AISC steel table. In this case, the W12X26 section is chosen. The term ST stands for standard single section.

```
* MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
MEMB RELEASE
1 5 10 14 15 18 26 20 TO 24 START MZ
4 9 13 14 15 18 25 19 21 TO 24 END MZ
```

The first set of members (1 5 10 etc) have local moment-z (MZ) released at the start joint. This means that these members cannot carry any moment-z (i.e., strong axis moment) at the start joint. The second set of members have MZ released at the end joints.

**Note:** Any line beginning with an asterisk (*) character is treated as a comment line.

```
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FT
```

Define the material properties for steel. The units are changed to inches and then back to feet in order to facilitate inputing the values in familiar units.

```
CONSTANT
MATERIAL STEEL ALL
```

The CONSTANT command instructs the program to use the defined steel material for all members.

```
SUPPORT
1 5 13 17 20 21 FIXED
```

A fixed support has been specified at the above joints.

```
LOADING 1 300 POUNDS PER SFT DL+LL
```
Load case 1 is initiated followed by a title.

FLOOR LOAD
YRANGE -0.5 0.5 FLOAD -0.30

All members within a range of -0.5 feet to 0.5 feet in the global Y direction (which is the entire floor of this model) are subjected to an floor load of 0.3 kips/sq.ft. The program converts area loads into individual member loads.

PERFORM ANALYSIS PRINT LOAD DATA

This command instructs the program to proceed with the analysis. The PRINT LOAD DATA command is specified to obtain a listing of the member loads which were generated from the FLOOR LOAD.

PARAMETERS
CODE AISC UNIFIED
BEAM 1 ALL
DMAX 2.0 ALL
DMIN 1.0 ALL
UNT 1.0 ALL
UNB 1.0 ALL

The PARAMETER command is used to specify steel design parameters (See Table 2.1 in Section 2.4 of the Technical Reference manual on page 1409). Design is to be performed per the specifications of the AISC ASD Code. The BEAM parameter is specified to perform design at every 1/12th point along the member length. DMAX and DMIN specify maximum and minimum depth limitations to be used during member selection. UNT and UNB are used to specify unsupported length for top and bottom flange to be used for calculation of allowable bending stress.

SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27

The above command instructs the program to select the most economical section from the British steel table for the members listed.

FINISH

The FINISH command terminates the STAAD run.
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FT
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 5 13 17 20 21 FIXED
LOADING 1 300 POUNDS PER SFT DL+LL
FLOOR LOAD
YRANGE -0.5 0.5 FLOAD -0.30
PERFORM ANALYSIS PRINT LOAD DATA
PARAMETERS
CODE AISC UNIFIED
BEAM 1.0 ALL
DMAX 2.0 ALL
DMIN 1.0 ALL
UNT 1.0 ALL
UNB 1.0 ALL
SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
FINISH

**STAAD Output File**

```
1. STAAD FLOOR  A FLOOR FRAME DESIGN WITH AREA LOAD
INPUT FILE: US-2 Area Load Generation on Floor Structure.STD
2. UNIT FT KIP
3. JOINT COORDINATES
4. 1 0. 0. 0. ; 5 20. 0. 0. ; 7 5. 0. 10.
5. 8 10. 0. 10. ; 9 13. 0. 10. ; 10 15. 0. 10. ; 11 16.5 0. 10.
6. 12 20. 0. 10. ; 13 0. 0. 25. ; 14 5. 0. 25. ; 15 11. 0. 25.
7. 16 16.5 0. 25 ; 17 20. 0. 25. 18 0. 0. 28.
8. 19 20. 0. 28. ; 20 0. 0. 35. ; 21 20. 0. 35.
9. MEMBER INCIDENCES
10. 1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
11. 15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
12. 19 7 14 ; 20 2 7 ; 21 9 15
13. 22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
```
**Application Examples**

**EX. American Design Examples**

```plaintext
14. 26 17 19 ; 27 12 17 ; 28 5 12
15. MEMB PROP AMERICAN
16. 1 TO 28 TABLE ST W12X26
17. * MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
18. MEMB RELEASE
19. 1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
20. 4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
21. UNIT INCHES
22. DEFINE MATERIAL START
23. ISOTROPIC STEEL
24. E 29000
25. POISSON 0.3
26. DENSITY 283E-006
27. ALPHA 6E-006
28. DAMP 0.03
29. TYPE STEEL
30. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
31. END DEFINE MATERIAL
32. UNIT FT
33. CONSTANT
34. MATERIAL STEEL ALL
35. SUPPORT
36. 1 5 13 17 20 21 FIXED
37. LOADING 1 300 POUNDS PER SFT DL+LL
38. FLOOR LOAD
    A FLOOR FRAME DESIGN WITH AREA LOAD          -- PAGE NO.  2
39. YRANGE -0.5 0.5 FLOAD -0.30

**NOTE** about Floor/OneWay Loads/Weights.  
Please note that depending on the shape of the floor you may  
have to break up the FLOOR/ONEWAY LOAD into multiple commands.  
For details please refer to Technical Reference Manual  
Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.

40. PERFORM ANALYSIS PRINT LOAD DATA

  PROBLEM STATISTICS
  ------------------------
  NUMBER OF JOINTS         20  NUMBER OF MEMBERS      28
  NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
  NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
  Using 64-bit analysis engine.  
  SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL PRIMARY LOAD CASES =  1, TOTAL DEGREES OF FREEDOM =  42
  TOTAL LOAD COMBINATION CASES =  0 SO FAR.
    A FLOOR FRAME DESIGN WITH AREA LOAD          -- PAGE NO.  3
  LOADING 1 300 POUNDS PER SFT DL+LL
  -------

  MEMBER LOAD - UNIT KIP FEET
  MEMBER  UDL  L1   L2  CON  L  LIN1  LIN2
          ------- ------- ------- ------- ------- -------
1        -0.0146 GY  0.21
1        -0.0439 GY  0.49
1        -0.0732 GY  0.79
1        -0.1025 GY  1.10
1        -0.1318 GY  1.41
1        -0.1611 GY  1.72
1        -0.1904 GY  2.04
1        -0.2197 GY  2.35
1         0.0000 GY  0.00
1         0.0000 GY  0.00
1        -0.0146 GY  0.21
```
<table>
<thead>
<tr>
<th></th>
<th>-0.7500 GY</th>
<th>2.50</th>
<th>10.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-0.1318 GY</td>
<td>3.59</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.1025 GY</td>
<td>3.90</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.0732 GY</td>
<td>4.21</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.0439 GY</td>
<td>4.51</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.0146 GY</td>
<td>4.79</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>-0.0146 GY</td>
<td>0.21</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>-0.0439 GY</td>
<td>0.49</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>-0.0732 GY</td>
<td>0.79</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>-0.1025 GY</td>
<td>1.10</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>-0.1318 GY</td>
<td>1.41</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>-0.1611 GY</td>
<td>1.72</td>
</tr>
<tr>
<td>20</td>
<td>-0.1904 GY</td>
<td>2.04</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.2197 GY</td>
<td>2.35</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.7500 GY</td>
<td>0.00</td>
<td>12.50</td>
</tr>
<tr>
<td>19</td>
<td>-0.2197 GY</td>
<td>12.65</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.1904 GY</td>
<td>12.96</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.1611 GY</td>
<td>13.28</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.1318 GY</td>
<td>13.59</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.1025 GY</td>
<td>13.90</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.0732 GY</td>
<td>14.21</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.0439 GY</td>
<td>14.51</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.0146 GY</td>
<td>14.79</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.0146 GY</td>
<td>0.21</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.0439 GY</td>
<td>0.49</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.0732 GY</td>
<td>0.79</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1025 GY</td>
<td>1.10</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1318 GY</td>
<td>1.41</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1611 GY</td>
<td>1.72</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1904 GY</td>
<td>2.04</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.2197 GY</td>
<td>2.35</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.2197 GY</td>
<td>2.65</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1904 GY</td>
<td>2.96</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1611 GY</td>
<td>3.28</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1318 GY</td>
<td>3.59</td>
<td></td>
</tr>
</tbody>
</table>

A FLOOR FRAME DESIGN WITH AREA LOAD

-- PAGE NO. 4
2                            -0.0439 GY   0.49
2                            -0.0732 GY   0.79
2                            -0.1025 GY   1.10
2                            -0.1318 GY   1.41
2                            -0.1611 GY   1.72
2                            -0.1904 GY   2.04
2                            -0.2197 GY   2.35
2                            -0.1904 GY   2.65
2                            -0.1611 GY   2.96
2                            -0.1318 GY   3.28
2                            -0.1025 GY   3.59
2                            -0.0732 GY   3.90
2                            -0.0439 GY   4.21
2                            -0.0146 GY   4.51
22                            -0.0146 GY   0.21
22                            -0.0439 GY   0.49
22                            -0.0732 GY   0.79
22                            -0.1025 GY   1.10
22                            -0.1318 GY   1.41
22                            -0.1611 GY   1.72
22                            -0.1904 GY   2.04
22                            -0.2197 GY   2.35
22    -0.7500 GY   2.50    7.50
22                            -0.2197 GY   7.65
22                            -0.1904 GY   7.96
22                            -0.1611 GY   8.28
22                            -0.1318 GY   8.59
22                            -0.1025 GY   8.90
22                            -0.0732 GY   9.21
22                            -0.0439 GY   9.51
22                            -0.0146 GY   9.79
5                            -0.0146 GY   0.21
5                            -0.0439 GY   0.49
A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    5
5                            -0.0732 GY   0.79
5                            -0.1025 GY   1.10
5                            -0.1318 GY   1.41
5                            -0.1611 GY   1.72
5                            -0.1904 GY   2.04
5                            -0.2197 GY   2.35
5                            -0.1904 GY   2.65
5                            -0.1611 GY   2.96
5                            -0.1318 GY   3.28
5                            -0.1025 GY   3.59
5                            -0.0732 GY   3.90
5                            -0.0439 GY   4.21
5                            -0.0146 GY   4.51
5                            -0.0146 GY   4.79
20                            -0.0146 GY   0.21
20                            -0.0439 GY   0.49
20                            -0.0732 GY   0.79
20                            -0.1025 GY   1.10
20                            -0.1318 GY   1.41
20                            -0.1611 GY   1.72
20                            -0.1904 GY   2.04
20                            -0.2197 GY   2.35
20    -0.7500 GY   2.50    7.50
|   | -0.2197 GY | 7.65 |
|   | -0.1904 GY | 7.96 |
|   | -0.1611 GY | 8.28 |
|   | -0.1318 GY | 8.59 |
|   | -0.1025 GY | 8.90 |
|   | -0.0732 GY | 9.21 |
|   | -0.0439 GY | 9.51 |
|   | -0.0146 GY | 9.79 |
|   | 0.0146 GY  | 0.21 |
|   | 0.0439 GY  | 0.49 |
|   | 0.0732 GY  | 0.79 |
|   | 0.1025 GY  | 1.10 |
|   | 0.1318 GY  | 1.41 |
|   | 0.1611 GY  | 1.72 |
|   | 0.1904 GY  | 2.04 |
|   | 0.2197 GY  | 2.35 |

A FLOOR FRAME DESIGN WITH AREA LOAD

-- PAGE NO. 6
<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>-0.0463</td>
<td>2.53</td>
</tr>
<tr>
<td>6</td>
<td>-0.0451</td>
<td>2.59</td>
</tr>
<tr>
<td>6</td>
<td>-0.0439</td>
<td>2.66</td>
</tr>
<tr>
<td>6</td>
<td>-0.0428</td>
<td>2.72</td>
</tr>
<tr>
<td>6</td>
<td>-0.0416</td>
<td>2.78</td>
</tr>
<tr>
<td>6</td>
<td>-0.0404</td>
<td>2.84</td>
</tr>
<tr>
<td>6</td>
<td>-0.0393</td>
<td>2.91</td>
</tr>
<tr>
<td>6</td>
<td>-0.0381</td>
<td>2.97</td>
</tr>
<tr>
<td>22</td>
<td>-0.0146</td>
<td>0.21</td>
</tr>
<tr>
<td>22</td>
<td>-0.0439</td>
<td>0.49</td>
</tr>
<tr>
<td>22</td>
<td>-0.0732</td>
<td>0.79</td>
</tr>
<tr>
<td>22</td>
<td>-0.1025</td>
<td>1.10</td>
</tr>
<tr>
<td>22</td>
<td>-0.1318</td>
<td>1.41</td>
</tr>
<tr>
<td>22</td>
<td>-0.1611</td>
<td>1.72</td>
</tr>
<tr>
<td>22</td>
<td>-0.1904</td>
<td>2.04</td>
</tr>
<tr>
<td>22</td>
<td>-0.2197</td>
<td>2.35</td>
</tr>
<tr>
<td>22</td>
<td>-0.7500</td>
<td>2.50</td>
</tr>
<tr>
<td>22</td>
<td>7.50</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-0.2197</td>
<td>7.65</td>
</tr>
<tr>
<td>4</td>
<td>-0.1984</td>
<td>7.96</td>
</tr>
<tr>
<td>4</td>
<td>-0.1611</td>
<td>8.28</td>
</tr>
<tr>
<td>4</td>
<td>-0.1318</td>
<td>8.59</td>
</tr>
<tr>
<td>4</td>
<td>-0.1025</td>
<td>8.90</td>
</tr>
<tr>
<td>4</td>
<td>-0.0732</td>
<td>9.21</td>
</tr>
<tr>
<td>4</td>
<td>-0.0439</td>
<td>9.51</td>
</tr>
<tr>
<td>4</td>
<td>-0.0146</td>
<td>9.79</td>
</tr>
<tr>
<td>4</td>
<td>-0.0146</td>
<td>0.21</td>
</tr>
<tr>
<td>4</td>
<td>-0.0439</td>
<td>0.49</td>
</tr>
<tr>
<td>4</td>
<td>-0.0732</td>
<td>0.79</td>
</tr>
<tr>
<td>4</td>
<td>-0.1025</td>
<td>1.10</td>
</tr>
<tr>
<td>4</td>
<td>-0.1318</td>
<td>1.41</td>
</tr>
<tr>
<td>4</td>
<td>-0.1611</td>
<td>1.72</td>
</tr>
<tr>
<td>4</td>
<td>-0.1904</td>
<td>2.04</td>
</tr>
<tr>
<td>4</td>
<td>-0.2197</td>
<td>2.35</td>
</tr>
</tbody>
</table>

**A FLOOR FRAME DESIGN WITH AREA LOAD**

--- PAGE NO.  7
<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>28</td>
<td>-0.0146 GY</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0586 GY</td>
<td>0.31</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0633 GY</td>
<td>0.44</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0680 GY</td>
<td>0.56</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0773 GY</td>
<td>0.69</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0820 GY</td>
<td>0.81</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0914 GY</td>
<td>0.94</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.2197 GY</td>
<td>1.15</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.1984 GY</td>
<td>1.46</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.1611 GY</td>
<td>1.78</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.1318 GY</td>
<td>2.09</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.1025 GY</td>
<td>2.40</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0732 GY</td>
<td>2.71</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0439 GY</td>
<td>3.01</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0146 GY</td>
<td>3.29</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0053 GY</td>
<td>0.12</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0158 GY</td>
<td>0.29</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0264 GY</td>
<td>0.47</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0369 GY</td>
<td>0.66</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0475 GY</td>
<td>0.85</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0580 GY</td>
<td>1.03</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0686 GY</td>
<td>1.22</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0791 GY</td>
<td>1.41</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0146 GY</td>
<td>0.21</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0439 GY</td>
<td>0.49</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0732 GY</td>
<td>0.79</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1025 GY</td>
<td>1.10</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1318 GY</td>
<td>1.41</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1611 GY</td>
<td>1.72</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0194 GY</td>
<td>2.04</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.2197 GY</td>
<td>2.35</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.2197 GY</td>
<td>2.50</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1984 GY</td>
<td>2.75</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1611 GY</td>
<td>3.00</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1318 GY</td>
<td>3.25</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1025 GY</td>
<td>3.50</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0732 GY</td>
<td>3.75</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0439 GY</td>
<td>4.00</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0146 GY</td>
<td>4.25</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.0628 GY</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.1884 GY</td>
<td>5.25</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.3140 GY</td>
<td>5.50</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.4396 GY</td>
<td>5.75</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.5652 GY</td>
<td>6.00</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.6909 GY</td>
<td>6.25</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.8165 GY</td>
<td>6.50</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.9421 GY</td>
<td>6.75</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.9162 GY</td>
<td>7.00</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.7940 GY</td>
<td>7.25</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.6719 GY</td>
<td>7.50</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.5497 GY</td>
<td>7.75</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.4276 GY</td>
<td>8.00</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.3054 GY</td>
<td>8.25</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.1832 GY</td>
<td>8.50</td>
<td></td>
</tr>
</tbody>
</table>

A FLOOR FRAME DESIGN WITH AREA LOAD -- PAGE NO. 8

---

**Application Examples**

**EX. American Design Examples**

STAAD.Pro

4373

User Manual
<table>
<thead>
<tr>
<th>Floor</th>
<th>GY 1</th>
<th>GY 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>0.0611</td>
<td>14.51</td>
</tr>
<tr>
<td>11</td>
<td>0.0649</td>
<td>0.29</td>
</tr>
<tr>
<td>11</td>
<td>0.1947</td>
<td>0.69</td>
</tr>
<tr>
<td>11</td>
<td>0.3245</td>
<td>1.12</td>
</tr>
<tr>
<td>11</td>
<td>0.4542</td>
<td>1.55</td>
</tr>
<tr>
<td>11</td>
<td>0.5840</td>
<td>1.99</td>
</tr>
<tr>
<td>11</td>
<td>0.7138</td>
<td>2.43</td>
</tr>
<tr>
<td>11</td>
<td>0.8436</td>
<td>2.87</td>
</tr>
<tr>
<td>11</td>
<td>0.9734</td>
<td>3.31</td>
</tr>
<tr>
<td>11</td>
<td>0.8400</td>
<td>3.68</td>
</tr>
<tr>
<td>11</td>
<td>0.5928</td>
<td>3.98</td>
</tr>
<tr>
<td>11</td>
<td>0.5016</td>
<td>4.29</td>
</tr>
<tr>
<td>11</td>
<td>0.4184</td>
<td>4.60</td>
</tr>
<tr>
<td>11</td>
<td>0.3192</td>
<td>4.91</td>
</tr>
<tr>
<td>11</td>
<td>0.2280</td>
<td>5.22</td>
</tr>
<tr>
<td>11</td>
<td>0.1368</td>
<td>5.52</td>
</tr>
<tr>
<td>11</td>
<td>0.0456</td>
<td>5.79</td>
</tr>
<tr>
<td>19</td>
<td>0.0590</td>
<td>0.60</td>
</tr>
<tr>
<td>19</td>
<td>0.1770</td>
<td>1.39</td>
</tr>
<tr>
<td>19</td>
<td>0.2950</td>
<td>2.26</td>
</tr>
<tr>
<td>19</td>
<td>0.4129</td>
<td>3.15</td>
</tr>
<tr>
<td>19</td>
<td>0.5309</td>
<td>4.03</td>
</tr>
<tr>
<td>19</td>
<td>0.6489</td>
<td>4.92</td>
</tr>
<tr>
<td>19</td>
<td>0.7669</td>
<td>5.82</td>
</tr>
<tr>
<td>19</td>
<td>0.8849</td>
<td>6.71</td>
</tr>
<tr>
<td>19</td>
<td>0.9734</td>
<td>7.62</td>
</tr>
<tr>
<td>19</td>
<td>0.8436</td>
<td>8.60</td>
</tr>
<tr>
<td>19</td>
<td>0.7138</td>
<td>9.58</td>
</tr>
<tr>
<td>19</td>
<td>0.5840</td>
<td>10.56</td>
</tr>
<tr>
<td>19</td>
<td>0.4542</td>
<td>11.54</td>
</tr>
<tr>
<td>19</td>
<td>0.3245</td>
<td>12.51</td>
</tr>
<tr>
<td>19</td>
<td>0.1947</td>
<td>13.47</td>
</tr>
<tr>
<td>6</td>
<td>0.0649</td>
<td>14.35</td>
</tr>
<tr>
<td>6</td>
<td>0.5049</td>
<td>6.18</td>
</tr>
<tr>
<td>6</td>
<td>0.4376</td>
<td>6.56</td>
</tr>
<tr>
<td>6</td>
<td>0.3703</td>
<td>6.93</td>
</tr>
<tr>
<td>6</td>
<td>0.3029</td>
<td>1.31</td>
</tr>
<tr>
<td>6</td>
<td>0.2356</td>
<td>1.68</td>
</tr>
<tr>
<td>6</td>
<td>0.1683</td>
<td>2.05</td>
</tr>
<tr>
<td>6</td>
<td>0.1018</td>
<td>2.42</td>
</tr>
<tr>
<td>6</td>
<td>0.0337</td>
<td>2.75</td>
</tr>
<tr>
<td>5</td>
<td>0.0590</td>
<td>8.29</td>
</tr>
<tr>
<td>5</td>
<td>0.1770</td>
<td>8.69</td>
</tr>
<tr>
<td>5</td>
<td>0.2950</td>
<td>1.12</td>
</tr>
<tr>
<td>5</td>
<td>0.4129</td>
<td>1.55</td>
</tr>
<tr>
<td>5</td>
<td>0.5309</td>
<td>1.99</td>
</tr>
<tr>
<td>5</td>
<td>0.6489</td>
<td>2.43</td>
</tr>
<tr>
<td>5</td>
<td>0.7669</td>
<td>2.87</td>
</tr>
<tr>
<td>5</td>
<td>0.8849</td>
<td>3.31</td>
</tr>
<tr>
<td>5</td>
<td>0.3873</td>
<td>3.62</td>
</tr>
<tr>
<td>5</td>
<td>0.3710</td>
<td>3.80</td>
</tr>
<tr>
<td>5</td>
<td>0.3547</td>
<td>3.98</td>
</tr>
<tr>
<td>5</td>
<td>0.3384</td>
<td>4.17</td>
</tr>
<tr>
<td>5</td>
<td>0.3221</td>
<td>4.35</td>
</tr>
<tr>
<td>5</td>
<td>0.3058</td>
<td>4.54</td>
</tr>
<tr>
<td>5</td>
<td>0.2895</td>
<td>4.72</td>
</tr>
<tr>
<td>5</td>
<td>0.2732</td>
<td>4.91</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th>Layer</th>
<th>GY</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>23</td>
<td>-0.0432</td>
<td>0.67</td>
</tr>
<tr>
<td>23</td>
<td>-0.1295</td>
<td>1.57</td>
</tr>
<tr>
<td>23</td>
<td>-0.2159</td>
<td>2.55</td>
</tr>
<tr>
<td>23</td>
<td>-0.3023</td>
<td>3.55</td>
</tr>
<tr>
<td>23</td>
<td>-0.3886</td>
<td>4.55</td>
</tr>
<tr>
<td>23</td>
<td>-0.4750</td>
<td>5.55</td>
</tr>
<tr>
<td>23</td>
<td>-0.5613</td>
<td>6.56</td>
</tr>
<tr>
<td>23</td>
<td>-0.6477</td>
<td>7.56</td>
</tr>
<tr>
<td>23</td>
<td>-0.5584</td>
<td>8.48</td>
</tr>
<tr>
<td>23</td>
<td>-0.4839</td>
<td>9.35</td>
</tr>
<tr>
<td>23</td>
<td>-0.4095</td>
<td>10.21</td>
</tr>
<tr>
<td>23</td>
<td>-0.3350</td>
<td>11.08</td>
</tr>
<tr>
<td>23</td>
<td>-0.2606</td>
<td>11.94</td>
</tr>
<tr>
<td>23</td>
<td>-0.1861</td>
<td>12.80</td>
</tr>
<tr>
<td>23</td>
<td>-0.1117</td>
<td>13.65</td>
</tr>
<tr>
<td>23</td>
<td>-0.0372</td>
<td>14.42</td>
</tr>
<tr>
<td>12</td>
<td>-0.0523</td>
<td>0.62</td>
</tr>
<tr>
<td>12</td>
<td>-0.1569</td>
<td>1.81</td>
</tr>
<tr>
<td>12</td>
<td>-0.2615</td>
<td>2.02</td>
</tr>
<tr>
<td>12</td>
<td>-0.3661</td>
<td>2.22</td>
</tr>
<tr>
<td>12</td>
<td>-0.4706</td>
<td>2.42</td>
</tr>
<tr>
<td>12</td>
<td>-0.5752</td>
<td>2.62</td>
</tr>
<tr>
<td>12</td>
<td>-0.6798</td>
<td>2.82</td>
</tr>
<tr>
<td>12</td>
<td>-0.7844</td>
<td>3.02</td>
</tr>
<tr>
<td>12</td>
<td>-0.5584</td>
<td>3.35</td>
</tr>
<tr>
<td>12</td>
<td>-0.4839</td>
<td>3.64</td>
</tr>
<tr>
<td>12</td>
<td>-0.4095</td>
<td>3.92</td>
</tr>
<tr>
<td>12</td>
<td>-0.3350</td>
<td>4.21</td>
</tr>
<tr>
<td>12</td>
<td>-0.2666</td>
<td>4.49</td>
</tr>
</tbody>
</table>

**A FLOOR FRAME DESIGN WITH AREA LOAD**

---

**STAAD.Pro**

4375

*User Manual*
### Application Examples

**EX. American Design Examples**

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>-0.0687 GY</td>
<td>0.24</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.1145 GY</td>
<td>0.38</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.1603 GY</td>
<td>0.53</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.2061 GY</td>
<td>0.69</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.2519 GY</td>
<td>0.84</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.2977 GY</td>
<td>0.99</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.3435 GY</td>
<td>1.14</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.3893 GY</td>
<td>1.26</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.4351 GY</td>
<td>1.36</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.4809 GY</td>
<td>1.46</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.5250 GY</td>
<td>1.56</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.1917 GY</td>
<td>1.66</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.2326 GY</td>
<td>1.75</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.1713 GY</td>
<td>1.85</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.1610 GY</td>
<td>1.95</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0072 GY</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0215 GY</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0359 GY</td>
<td>0.55</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0502 GY</td>
<td>0.77</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0646 GY</td>
<td>0.99</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0790 GY</td>
<td>1.21</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0933 GY</td>
<td>1.42</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.1077 GY</td>
<td>1.64</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.1077 GY</td>
<td>1.86</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0933 GY</td>
<td>2.08</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0790 GY</td>
<td>2.29</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0646 GY</td>
<td>2.51</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0502 GY</td>
<td>2.73</td>
<td></td>
</tr>
</tbody>
</table>

**A FLOOR FRAME DESIGN WITH AREA LOAD**

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>-0.0359 GY</td>
<td>2.95</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0215 GY</td>
<td>3.16</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-0.0072 GY</td>
<td>3.35</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0072 GY</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0215 GY</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0359 GY</td>
<td>0.55</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0502 GY</td>
<td>0.77</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0646 GY</td>
<td>0.99</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0790 GY</td>
<td>1.21</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0933 GY</td>
<td>1.42</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.1077 GY</td>
<td>1.64</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.5250 GY</td>
<td>1.75</td>
<td>13.25</td>
</tr>
<tr>
<td>27</td>
<td>-0.1077 GY</td>
<td>13.36</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0933 GY</td>
<td>13.58</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0790 GY</td>
<td>13.79</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0646 GY</td>
<td>14.01</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0502 GY</td>
<td>14.23</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0359 GY</td>
<td>14.45</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0215 GY</td>
<td>14.66</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0072 GY</td>
<td>14.85</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0072 GY</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0215 GY</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0359 GY</td>
<td>0.55</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0502 GY</td>
<td>0.77</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0646 GY</td>
<td>0.99</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0790 GY</td>
<td>1.21</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0933 GY</td>
<td>1.42</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.1077 GY</td>
<td>1.64</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.1077 GY</td>
<td>1.86</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples
EX. American Design Examples

<table>
<thead>
<tr>
<th>Floor</th>
<th>Load Factor</th>
<th>Load</th>
<th>Load Factor</th>
<th>Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>-0.0933 GY</td>
<td>2.08</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0790 GY</td>
<td>2.29</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0646 GY</td>
<td>2.51</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0502 GY</td>
<td>2.73</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0359 GY</td>
<td>2.95</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0215 GY</td>
<td>3.16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0072 GY</td>
<td>3.35</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0072 GY</td>
<td>0.15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0215 GY</td>
<td>0.34</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0359 GY</td>
<td>0.55</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0502 GY</td>
<td>0.77</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0646 GY</td>
<td>0.99</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0790 GY</td>
<td>1.21</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0933 GY</td>
<td>1.42</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.1077 GY</td>
<td>1.64</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.5250 GY</td>
<td>1.75</td>
<td>13.25</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.1077 GY</td>
<td>13.36</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0933 GY</td>
<td>13.58</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0790 GY</td>
<td>13.79</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0646 GY</td>
<td>14.01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0502 GY</td>
<td>14.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0359 GY</td>
<td>14.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0215 GY</td>
<td>14.66</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.0072 GY</td>
<td>14.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0053 GY</td>
<td>0.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0158 GY</td>
<td>0.29</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0264 GY</td>
<td>0.47</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

A FLOOR FRAME DESIGN WITH AREA LOAD

<table>
<thead>
<tr>
<th>Floor</th>
<th>Load Factor</th>
<th>Load</th>
<th>Load Factor</th>
<th>Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>26</td>
<td>-0.0369 GY</td>
<td>0.66</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0475 GY</td>
<td>0.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0580 GY</td>
<td>1.03</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0686 GY</td>
<td>1.22</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0791 GY</td>
<td>1.41</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0791 GY</td>
<td>1.59</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0686 GY</td>
<td>1.78</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0580 GY</td>
<td>1.97</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0475 GY</td>
<td>2.15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0369 GY</td>
<td>2.34</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0264 GY</td>
<td>2.53</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0158 GY</td>
<td>2.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0053 GY</td>
<td>2.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0053 GY</td>
<td>0.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0158 GY</td>
<td>0.29</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0264 GY</td>
<td>0.47</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0369 GY</td>
<td>0.66</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0475 GY</td>
<td>0.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0580 GY</td>
<td>1.03</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0686 GY</td>
<td>1.22</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0791 GY</td>
<td>1.41</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.4500 GY</td>
<td>1.50</td>
<td>18.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0791 GY</td>
<td>18.59</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0686 GY</td>
<td>18.78</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0580 GY</td>
<td>18.97</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0475 GY</td>
<td>19.15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0369 GY</td>
<td>19.34</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0264 GY</td>
<td>19.52</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0158 GY</td>
<td>19.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0053 GY</td>
<td>19.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Layer</td>
<td>Area Load</td>
<td>GY 1</td>
<td>GY 2</td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>-----------</td>
<td>------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.4500 GY</td>
<td>1.50</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-0.4500 GY</td>
<td>6.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-0.4500 GY</td>
<td>5.50</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.4500 GY</td>
<td>2.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-1.0500 GY</td>
<td>3.50</td>
<td>16.50</td>
<td></td>
</tr>
</tbody>
</table>

A FLOOR FRAME DESIGN WITH AREA LOAD

---

---
<table>
<thead>
<tr>
<th>Layer</th>
<th>Gyration Radius</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>-0.2584</td>
<td>1.98</td>
</tr>
<tr>
<td>25</td>
<td>-0.3158</td>
<td>2.41</td>
</tr>
<tr>
<td>25</td>
<td>-0.3732</td>
<td>2.85</td>
</tr>
<tr>
<td>25</td>
<td>-0.4307</td>
<td>3.29</td>
</tr>
<tr>
<td>25</td>
<td>-0.4307</td>
<td>3.71</td>
</tr>
<tr>
<td>25</td>
<td>-0.3732</td>
<td>4.15</td>
</tr>
<tr>
<td>25</td>
<td>-0.3158</td>
<td>4.59</td>
</tr>
<tr>
<td>25</td>
<td>-0.2584</td>
<td>5.02</td>
</tr>
<tr>
<td>25</td>
<td>-0.2010</td>
<td>5.46</td>
</tr>
<tr>
<td>25</td>
<td>-0.1436</td>
<td>5.89</td>
</tr>
<tr>
<td>25</td>
<td>-0.0861</td>
<td>6.32</td>
</tr>
<tr>
<td>25</td>
<td>-0.0287</td>
<td>6.71</td>
</tr>
<tr>
<td>15</td>
<td>-0.0287</td>
<td>0.29</td>
</tr>
<tr>
<td>15</td>
<td>-0.0861</td>
<td>0.68</td>
</tr>
<tr>
<td>15</td>
<td>-0.1436</td>
<td>1.11</td>
</tr>
<tr>
<td>15</td>
<td>-0.2010</td>
<td>1.54</td>
</tr>
<tr>
<td>15</td>
<td>-0.2584</td>
<td>1.98</td>
</tr>
<tr>
<td>15</td>
<td>-0.3158</td>
<td>2.41</td>
</tr>
<tr>
<td>15</td>
<td>-0.3732</td>
<td>2.85</td>
</tr>
<tr>
<td>15</td>
<td>-0.4307</td>
<td>3.29</td>
</tr>
<tr>
<td>15</td>
<td>-1.0500</td>
<td>3.50</td>
</tr>
<tr>
<td>15</td>
<td>-0.4307</td>
<td>16.71</td>
</tr>
<tr>
<td>15</td>
<td>-0.3732</td>
<td>17.15</td>
</tr>
<tr>
<td>15</td>
<td>-0.3158</td>
<td>17.59</td>
</tr>
<tr>
<td>15</td>
<td>-0.2584</td>
<td>18.02</td>
</tr>
<tr>
<td>15</td>
<td>-0.2010</td>
<td>18.46</td>
</tr>
<tr>
<td>15</td>
<td>-0.1436</td>
<td>18.89</td>
</tr>
<tr>
<td>15</td>
<td>-0.0861</td>
<td>19.32</td>
</tr>
<tr>
<td>15</td>
<td>-0.0287</td>
<td>19.71</td>
</tr>
<tr>
<td>16</td>
<td>-0.0287</td>
<td>0.29</td>
</tr>
<tr>
<td>16</td>
<td>-0.0861</td>
<td>0.68</td>
</tr>
<tr>
<td>16</td>
<td>-0.1436</td>
<td>1.11</td>
</tr>
<tr>
<td>16</td>
<td>-0.2010</td>
<td>1.54</td>
</tr>
<tr>
<td>16</td>
<td>-0.2584</td>
<td>1.98</td>
</tr>
<tr>
<td>16</td>
<td>-0.3158</td>
<td>2.41</td>
</tr>
<tr>
<td>16</td>
<td>-0.3732</td>
<td>2.85</td>
</tr>
<tr>
<td>16</td>
<td>-0.4307</td>
<td>3.29</td>
</tr>
<tr>
<td>16</td>
<td>-0.4307</td>
<td>3.71</td>
</tr>
<tr>
<td>16</td>
<td>-0.3732</td>
<td>4.15</td>
</tr>
<tr>
<td>16</td>
<td>-0.3158</td>
<td>4.59</td>
</tr>
<tr>
<td>16</td>
<td>-0.2584</td>
<td>5.02</td>
</tr>
<tr>
<td>16</td>
<td>-0.2010</td>
<td>5.46</td>
</tr>
<tr>
<td>16</td>
<td>-0.1436</td>
<td>5.89</td>
</tr>
<tr>
<td>16</td>
<td>-0.0861</td>
<td>6.32</td>
</tr>
<tr>
<td>16</td>
<td>-0.0287</td>
<td>6.71</td>
</tr>
</tbody>
</table>

************ END OF DATA FROM INTERNAL STORAGE ************

41. PARAMETERS
42. CODE AISC UNIFIED
43. BEAM 1.0 ALL
44. DMAX 2.0 ALL
45. DMIN 1.0 ALL
46. UNT 1.0 ALL
47. UNB 1.0 ALL
48. SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27

STEEL DESIGN

A FLOOR FRAME DESIGN WITH AREA LOAD

STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.1)
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

*** NOTE : AISC 360-16 Design Statement for STAAD.Pro. ***

*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the notations and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16 design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING:

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis. E.g : Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
F-L-B = Flange Local Buckling
W-L-B = Web Local Buckling
L-L-B = Leg Local Buckling
C-F-Y = Compression Flange Yielding
T-F-Y = Tension Flange Yielding

--- PAGE NO. 16 ---

STAAD.Pro Member Selection - AISC 360-16 LRFD (V1.1)

--- PAGE NO. 16 ---

STAAD.Pro Member Selection - AISC 360-16 LRFD (V1.1)
<table>
<thead>
<tr>
<th>Member No</th>
<th>Profile</th>
<th>Status</th>
<th>Ratio</th>
<th>Loadcase</th>
<th>Location</th>
<th>Pz</th>
<th>T</th>
<th>Vy</th>
<th>Vx</th>
<th>Tz</th>
<th>Mx</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>ST W16X26</td>
<td>PASS</td>
<td>0.862</td>
<td>1</td>
<td>3.00</td>
<td>0.000</td>
<td>T</td>
<td>3.529</td>
<td>0.000</td>
<td>0.6495E-02</td>
<td>-102.8</td>
</tr>
<tr>
<td>11</td>
<td>ST W18X35</td>
<td>PASS</td>
<td>0.960</td>
<td>1</td>
<td>3.50</td>
<td>0.000</td>
<td>T</td>
<td>0.4731</td>
<td>0.000</td>
<td>0.7231E-03</td>
<td>-172.4</td>
</tr>
<tr>
<td>14</td>
<td>ST W12X22</td>
<td>PASS</td>
<td>0.919</td>
<td>1</td>
<td>10.00</td>
<td>0.000</td>
<td>T</td>
<td>0.000</td>
<td>0.000</td>
<td>0.2891E-18</td>
<td>-72.69</td>
</tr>
<tr>
<td>15</td>
<td>ST W12X19</td>
<td>PASS</td>
<td>0.755</td>
<td>1</td>
<td>10.00</td>
<td>0.000</td>
<td>T</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>-50.36</td>
</tr>
<tr>
<td>16</td>
<td>ST W12X16</td>
<td>PASS</td>
<td>0.580</td>
<td>1</td>
<td>0.00</td>
<td>0.000</td>
<td>T</td>
<td>2.663</td>
<td>0.000</td>
<td>0.000</td>
<td>-31.50</td>
</tr>
<tr>
<td>18</td>
<td>ST W14X22</td>
<td>PASS</td>
<td>0.645</td>
<td>1</td>
<td>12.50</td>
<td>0.000</td>
<td>T</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>-57.81</td>
</tr>
<tr>
<td>19</td>
<td>ST W21X44</td>
<td>PASS</td>
<td>0.861</td>
<td>1</td>
<td>0.00</td>
<td>0.000</td>
<td>T</td>
<td>5.183</td>
<td>0.000</td>
<td>0.4163E-02</td>
<td>-221.8</td>
</tr>
</tbody>
</table>
- Member :  21

Member No:       21       Profile:  ST  W12X16              (AISC SECTIONS)
Status:        PASS       Ratio:         0.608       Loadcase:        1
Location:      7.57       Ref:      Cl.F2.1                                
Pz:       0.000     T     Vy:       -.1907           Vx:      0.000        
Tz:      0.1216E-01       My:        0.000           Mx:     -32.99        

- Member :    23

Member No:       23       Profile:  ST  W12X16              (AISC SECTIONS)
Status:        PASS       Ratio:         0.505       Loadcase:        1
Location:      7.50       Ref:      Cl.F2.1                                
Pz:       0.000     T     Vy:       0.3936           Vx:      0.000        
Tz:      -.8275E-03       My:        0.000           Mx:     -27.39        

- Member :    24

Member No:       24       Profile:  ST  W12X16              (AISC SECTIONS)
Status:        PASS       Ratio:         0.317       Loadcase:        1
Location:      5.00       Ref:      Cl.F2.1                                
Pz:       0.000     T     Vy:       0.9537E-06       Vx:      0.000        
Tz:      -.1067E-01       My:        0.000           Mx:     -17.19        

- Member :    27

Member No:       27       Profile:  ST  W18X35              (AISC SECTIONS)
Status:        PASS       Ratio:         0.952       Loadcase:        1
Location:      0.00       Ref:      Cl.F2.1                                
Pz:       0.000     T     Vy:       -7.917           Vx:      0.000        
Tz:       0.000           My:        0.000           Mx:     -170.9        

49. FINISH

*******************************************************************************
**WARNING**  SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
            IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
            ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
            TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
            FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
            FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
            COMMANDS.
*******************************************************************************
EX. US-3 Soil Springs for Portal Frame

A portal frame type steel structure is sitting on concrete footings. The soil is to be considered as an elastic foundation. The value of soil subgrade reaction is known from which spring constants are calculated by multiplying the subgrade reaction by the tributary area of each modeled spring.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-3 Soil Springs for Portal Frame.STD when you install the program.

![Diagram of portal frame](image)

*Figure 456: Example Problem No. 3*

Where:

- \( B = 20 \text{ ft}, \ H = 10 \text{ ft}, \ F_1 = 4 \text{ ft}, \ F_2 = 8 \text{ ft} \)
- \( P = 5 \text{ kips}, \ w = 3 \text{ kips/ft} \)
- Soil Subgrade Reaction = 250 kips/ft\(^3\)
Table 656: Spring constant calculation

<table>
<thead>
<tr>
<th>Springs of Joints</th>
<th>Spring Constant</th>
</tr>
</thead>
<tbody>
<tr>
<td>1, 5, 10, &amp; 14 (Edges)</td>
<td>8 x 1 x 250 =</td>
</tr>
<tr>
<td></td>
<td>2,000 kips/ft</td>
</tr>
<tr>
<td>2, 3, 4, 11, 12, &amp; 13 (Interior)</td>
<td>8 x 2 x 250 =</td>
</tr>
<tr>
<td></td>
<td>4,000 kips/ft</td>
</tr>
</tbody>
</table>

Actual input is shown in bold lettering followed by explanation.

**STAAD PLANE PORTAL ON FOOTING FOUNDATION**

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

**UNIT FT KIPS**

 Defines the input units for the data that follows.

**JOINT COORDINATES**

<table>
<thead>
<tr>
<th>Joint Number</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>5</td>
<td>8.0</td>
<td>0.0</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>4.0</td>
<td>10.0</td>
<td>0.0</td>
<td>7</td>
<td>4.0</td>
<td>20.0</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>24.0</td>
<td>20.0</td>
<td>0.0</td>
<td>9</td>
<td>24.0</td>
<td>10.0</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>20.0</td>
<td>0.0</td>
<td>0.0</td>
<td>14</td>
<td>28.0</td>
<td>0.0</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

**MEMBER INCIDENCES**

<table>
<thead>
<tr>
<th>Member</th>
<th>Joint 1</th>
<th>Joint 2</th>
<th>Joint 3</th>
<th>Joint 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>2</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>12</td>
</tr>
<tr>
<td>11</td>
<td>10</td>
<td>11</td>
<td>14</td>
<td></td>
</tr>
</tbody>
</table>

Defines the members by the joints to which they are connected.

**MEMBER PROPERTIES AMERICAN**

<table>
<thead>
<tr>
<th>Member</th>
<th>Joint 1</th>
<th>Joint 2</th>
<th>Joint 3</th>
<th>Joint 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4</td>
<td>11</td>
<td>14</td>
<td>PRIS</td>
</tr>
<tr>
<td></td>
<td>YD 1.0</td>
<td>ZD 8.0</td>
<td></td>
<td>PRIS</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>12</td>
<td>13</td>
<td>YD 2.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>ZD 8.0</td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>9</td>
<td>10</td>
<td>TA</td>
</tr>
<tr>
<td></td>
<td>ST</td>
<td>W10X33</td>
<td></td>
<td>ST</td>
</tr>
<tr>
<td>7</td>
<td>8</td>
<td></td>
<td></td>
<td>W12X26</td>
</tr>
</tbody>
</table>

The first two lines define member properties as prismatic (PRIS) followed by depth (YD) and width (ZD) values. The program will calculate the properties necessary to perform the analysis. See G.6.1 Prismatic Properties (on page 2323) for additional information. Member properties for the remaining members are chosen from the British steel table. The term ST stands for standard single section.

* E FOR STEEL IS 29,000 AND FOR CONCRETE 3000
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
The CONSTANT command initiates input for material constants like modulus of elasticity, Density, and Poisson's ratio. The length unit is changed from FT to INCH to facilitate the input in familiar units.

**Note:** Any line beginning with an asterisk (*) character is treated as a comment line.

The supports for the structure are specified above. The first set of joints are supports restrained in all directions except global moment-z (MZ). Also, a spring having a spring constant of 4,000 kip/ft is provided in the global Y direction at these nodes. The second set is similar to the former except for a different value of the spring constant.

Load case 1 is initiated followed by a title. The selfweight of the structure is specified as acting in the global Y direction with a -1.0 factor. Since global Y is vertically upwards, the -1.0 factor indicates that this load will act downwards.

Load 1 contains joint loads also. FX indicates that the load is a force in the global X direction.

Load 1 contains member loads also. GY indicates that the load acts in the global Y direction. The term UNI stands for uniformly distributed load, and is applied on members 7 and 8, acting downwards.

This command instructs the program to proceed with the analysis.
The above PRINT command instructs the program to print analysis results which include joint displacements, member forces and support reactions.

FINISH

This command terminates the STAAD run.

Input File

STAAD PLANE PORTAL ON FOOTING FOUNDATION
UNIT FT KIPS
JOINT COORDINATES
1 0.0 0.0 0.0 5 8.0 0.0 0.0
6 4.0 10.0 0.0 ; 7 4.0 20.0 0.0
8 24.0 20.0 0.0 ; 9 24.0 10.0 0.0
10 20.0 0.0 0.0 14 28.0 0.0 0.0
MEMBER INCIDENCES
1 1 2 4
5 3 6 ; 6 6 7
7 7 8 ; 8 6 9
9 8 9 ; 10 9 12
11 10 11 14
MEMBER PROPERTIES AMERICAN
1 4 11 14 PRIS YD 1.0 ZD 8.0
2 3 12 13 PRIS YD 2.0 ZD 8.0
5 6 9 10 TA ST W10X33
7 8 TA ST W12X26
* E FOR STEEL IS 29,000 AND FOR CONCRETE 3000
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.283e-003
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3000
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 5 TO 10
MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
UNIT FT
SUPPORTS
2 TO 4 11 TO 13 FIXED BUT MZ KFY 4000.
1 5 10 14 FIXED BUT MZ KFY 2000.
LOADING 1 DEAD AND WIND LOAD COMBINED
SELF Y -1.0
JOINT LOAD
6 7 FX 5.0
MEMBER LOAD
7 8 UNI GY -3.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH

STAAD Output File

1. STAAD PLANE PORTAL ON FOOTING FOUNDATION
INPUT FILE: US-3 Soil Springs for Portal Frame.STD
2. UNIT FT KIPS
3. JOINT COORDINATES
4. 1 0.0 0.0 0.0 5 8.0 0.0 0.0
5. 6 4.0 10.0 0.0 ; 7 4.0 20.0 0.0
6. 8 24.0 20.0 0.0 ; 9 24.0 10.0 0.0
7. 10 20.0 0.0 0.0 14 28.0 0.0 0.0
8. MEMBER INCIDENCES
9. 1 1 2 4
10. 5 3 6 ; 6 67
11. 7 7 8 ; 8 69
12. 9 8 9 ; 10 9 12
13. 11 10 11 14
14. MEMBER PROPERTIES AMERICAN
15. 1 4 11 14 PRIS YD 1.0 ZD 8.0
16. 2 3 12 13 PRIS YD 2.0 ZD 8.0
17. 5 6 9 10 TA ST W10X33
18. 7 8 TA ST W12X26
19. * E FOR STEEL IS 29,000 AND FOR CONCRETE 3000
20. UNIT INCHES
21. DEFINE MATERIAL START
22. ISOTROPIC STEEL
23. E 29000
24. POISSON 0.3
25. DENSITY 0.283E-003
26. ALPHA 6E-006
27. DAMP 0.03
28. TYPE STEEL
29. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
30. ISOTROPIC CONCRETE
31. E 3000
32. POISSON 0.17
33. DENSITY 8.68E-005
34. ALPHA 5E-006
35. DAMP 0.05
36. G 1346.15
37. TYPE CONCRETE
38. STRENGTH FCU 4  
PORTAL ON FOOTING FOUNDATION                               -- PAGE NO.  2
39. END DEFINE MATERIAL
40. CONSTANTS
41. MATERIAL STEEL MEMB 5 TO 10
42. MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
43. UNIT FT
44. SUPPORTS
45. 2 TO 4 11 TO 13 FIXED BUT MZ KFY 4000.
46. 1 5 10 14 FIXED BUT MZ KFY 2000.
47. LOADING 1 DEAD AND WIND LOAD COMBINED
48. SELF Y -1.0
49. JOINT LOAD
50. 6 7 FX 5.0
51. MEMBER LOAD
52. 7 8 UNI GY -3.0
53. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------------------
NUMBER OF JOINTS         14  NUMBER OF MEMBERS      14
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     10
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      32
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
54. PRINT ANALYSIS RESULTS

ANALYSIS RESULTS
PORTAL ON FOOTING FOUNDATION                               -- PAGE NO.  3
JOINT DISPLACEMENT (INCH RADIANS)STRUCTURE TYPE = PLANE
------------------
JOINT LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
1    1  0.00000  -0.04256   0.00000   0.00000   0.00000  -0.00028
2    1  0.00000  -0.04892   0.00000   0.00000   0.00000  -0.00023
3    1  0.00000  -0.05439   0.00000   0.00000   0.00000  -0.00017
4    1  0.00000  -0.05854   0.00000   0.00000   0.00000  -0.00010
5    1  0.00000  -0.06134   0.00000   0.00000   0.00000  -0.00004
6    1  0.32187  -0.07856   0.00000   0.00000   0.00000  -0.00483
7    1  0.64023  -0.09085   0.00000   0.00000   0.00000  -0.00669
8    1  0.62253  -0.10247   0.00000   0.00000   0.00000  -0.00393
9    1  0.32940  -0.08883   0.00000   0.00000   0.00000  -0.00014
10   1  0.00000  -0.03597   0.00000   0.00000   0.00000  -0.00055
11   1  0.00000  -0.04884   0.00000   0.00000   0.00000  -0.00051
12   1  0.00000  -0.06099   0.00000   0.00000   0.00000  -0.00049
13   1  0.00000  -0.07163   0.00000   0.00000   0.00000  -0.00043
14   1  0.00000  -0.08045   0.00000   0.00000   0.00000  -0.00035

PORTAL ON FOOTING FOUNDATION                               -- PAGE NO.  4
SUPPORT REACTIONS -UNIT KIPS FEET  STRUCTURE TYPE = PLANE
-----------------
JOINT LOAD   FORCE-X   FORCE-Y   FORCE-Z   MOM-X   MOM-Y   MOM Z
2    1  0.00   16.31      0.00      0.00      0.00      0.00
3    1  0.60   18.13      0.00      0.00      0.00      0.00
4    1  0.00   19.51      0.00      0.00      0.00      0.00
11   1  0.00   16.28      0.00      0.00      0.00      0.00
12   1  0.40   20.33      0.00      0.00      0.00      0.00
13   1  0.00   23.88      0.00      0.00      0.00      0.00
1    1  0.00    7.09      0.00      0.00      0.00      0.00
5    1  0.00  10.22      0.00      0.00      0.00      0.00
### Portal on Footing Foundation

#### Member End Forces

**Structure Type = Plane**

---

All units are -- KIPS FEET (Local)

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>7.09</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>-4.69</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>11.79</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>2</td>
<td>0.00</td>
<td>21.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-11.79</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>2</td>
<td>0.00</td>
<td>-16.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>18.05</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>3</td>
<td>56.87</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>18.93</td>
<td>-22.54</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>3</td>
<td>-56.54</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-12.95</td>
<td>-50.40</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>6</td>
<td>29.01</td>
<td>-11.36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-44.54</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>6</td>
<td>-16.36</td>
<td>31.84</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-117.94</td>
</tr>
<tr>
<td>9</td>
<td>9</td>
<td>8</td>
<td>31.84</td>
<td>16.36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>94.86</td>
</tr>
<tr>
<td>10</td>
<td>10</td>
<td>9</td>
<td>65.16</td>
<td>9.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>49.17</td>
</tr>
<tr>
<td>11</td>
<td>11</td>
<td>10</td>
<td>-65.49</td>
<td>-9.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>44.85</td>
</tr>
<tr>
<td>12</td>
<td>12</td>
<td>10</td>
<td>0.00</td>
<td>5.99</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>13</td>
<td>13</td>
<td>11</td>
<td>0.00</td>
<td>-3.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>9.59</td>
</tr>
<tr>
<td>14</td>
<td>14</td>
<td>11</td>
<td>0.00</td>
<td>19.87</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-9.59</td>
</tr>
<tr>
<td>15</td>
<td>15</td>
<td>12</td>
<td>0.00</td>
<td>-15.87</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>44.54</td>
</tr>
<tr>
<td>16</td>
<td>16</td>
<td>12</td>
<td>0.00</td>
<td>-30.88</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-7.95</td>
</tr>
<tr>
<td>17</td>
<td>17</td>
<td>12</td>
<td>0.00</td>
<td>34.88</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>24.42</td>
</tr>
<tr>
<td>18</td>
<td>18</td>
<td>13</td>
<td>0.00</td>
<td>-11.01</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-24.42</td>
</tr>
<tr>
<td>19</td>
<td>19</td>
<td>13</td>
<td>0.00</td>
<td>32.99</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>68.77</td>
</tr>
</tbody>
</table>

---

************** END OF LATEST ANALYSIS RESULT **************

55. FINISH

*********** END OF THE STAAD.Pro RUN ***********

**** DATE= APR 14,2019 TIME= 22:57: 6 ****

*******************************************************************************
* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/ *
* *
* Details about additional assistance from *
* Bentley and Partners can be found at program menu *
* Help->Technical Support *
* *
* Copyright (c) 1997-2017 Bentley Systems, Inc. *
* http://www.bentley.com *
*******************************************************************************

### Related Links

- [M. To assign a spring support](#) (on page 814)
- [TR.27.1 Global Support Specification](#) (on page 2514)
- [Create Support dialog](#) (on page 2983)
EX. US-4 Inactive Members in a Braced Frame

This example is a typical case of a load-dependent structure where the structural condition changes for different load cases. In this example, different bracing members are made inactive for different load cases. This is done to prevent these members from carrying any compressive forces.

This problem is installed with the program by default to 
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-4 Inactive Members in a Braced Frame.STD when you install the program.

Where:

\[ L_1 = 180 \text{ in.}, L_2 = 240 \text{ in.} \]
\[ P_1 = 15 \text{ kips}, P_2 = 30 \text{ kips} \]

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE
* A PLANE FRAME STRUCTURE WITH TENSION BRACING
```
Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

UNIT INCH KIP

Defines the input units for the data that follows.

SET NL 3

This structure has to be analyzed for three primary load cases. Consequently, the modeling of our problem requires us to define three sets of data, with each set containing a load case and an associated analysis command. Also, the members which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have two commands, one called SET NL and the other called CHANGE. The SET NL command is used above to indicate the total number of primary load cases that the file contains. The CHANGE command will come in later (after the PERFORM ANALYSIS command).

JOINT COORDINATES
1 0 0 0 3
4 0 180. 0 6
7 240. 360. 0

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCE
1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
13 6 7 ; 14 5 8

Defines the members by the joints to which they are connected.

MEMBER TRUSS
9 TO 14

The preceding command defines that members 9 through 14 are of type truss. This means these members can only carry axial tension/compression and no moments.

MEMBER PROP AMERICAN
1 TO 5 TABLE ST W12X26
6  7  8 TA ST W18X35
9 TO 14 TA LD L50505

Properties for all members are assigned from the American (AISC) steel table. The word ST stands for standard single section. The word LD stands for long leg back-to-back double angle. Since the spacing between the two angles of the double angle is not provided, it is assumed to be 0.0.

DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
Define the material properties for steel. The units are changed to inches and then back to feet in order to facilitate inputing the values in familiar units.

```
CONSTANTS
MATERIAL STEEL ALL
```

The CONSTANT command initiates input for material definitions.

```
SUPPORT
1 2 3 PINNED
```

Pinned supports are specified at Joints 1, 2 and 3. The word PINNED signifies that no moments will be carried by these supports.

```
INACTIVE MEMBERS 9 TO 14
```

The preceding command makes the listed members inactive. The stiffness contribution of these members will not be considered in the analysis till they are made active again.

```
UNIT FT
LOADING 1 DEAD AND LIVE LOAD
```

Load case 1 is initiated followed by a title. The length UNIT is changed from INCH to FT for input values which follow.

```
MEMBER LOAD
6 8 UNI GY -1.0
7 UNI GY -1.5
```

Load 1 contains member loads. GY indicates that the load acts in the global Y direction. The word UNI stands for uniformly distributed load. The load is applied on members 6, 7, and 8.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. It is worth noting that members 9 to 14 will not be used in this analysis since they were declared inactive earlier. In other words, for dead and live load, the bracing members are not used to carry any load.

```
CHANGES
```

The members inactivated earlier are restored using the CHANGE command.

```
INACTIVE MEMBERS 10 11 13
```

A new set of members are made inactive. The stiffness contribution from these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

```
LOADING 2 WIND FROM LEFT
```

Load case 2 is initiated followed by a title.

```
JOINT LOAD
4 FX 30 ; 7 FX 15
```

Load 2 contains joint loads. FX indicates that the load is a force in the global X direction. Nodes 4 and 7 are subjected to the loads.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. The analysis will be performed for load case 2 only.
The above CHANGE command is an instruction to re-activate all inactive members.

| INACTIVE MEMBERS 9 12 14 |

Members 9, 12 and 14 are made inactive. The stiffness contribution of these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

| LOADING 3 WIND FROM RIGHT |

Load case 3 is initiated followed by a title.

| JOINT LOAD |
| 6 FX -30 ; 8 FX -15 |

Load 3 contains joint loads at nodes 6 and 8. FX indicates that the load is a force in the global X direction. The negative numbers (-30 and -15) indicate that the load is acting along the negative global X direction.

| LOAD COMBINATION 4 |
| 1 0.75 2 0.75 |
| LOAD COMBINATION 5 |
| 1 0.75 3 0.75 |

Load combination case 4 involves the algebraic summation of the results of load cases 1 and 2 after multiplying each by a factor of 0.75. For load combinations, the program simply gathers the results of the component primary cases, factors them appropriately, and combines them algebraically. Thus, an analysis in the real sense of the term (multiplying the inverted stiffness matrix by the load vector) is not carried out for load combination cases. Load combination case 5 combines the results of load cases 1 and 3.

| PERFORM ANALYSIS |

This command instructs the program to proceed with the analysis. Only primary load case 3 will be considered for this analysis. (As explained earlier, a combination case is not truly analyzed for, but handled using other means.)

| CHANGE |

The above CHANGE command will re-activate all inactive members.

| LOAD LIST ALL |

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the "active" load cases. The LOAD LIST ALL command enables all the load cases in the structure to be made active for further processing.

| PRINT MEMBER FORCES |

The preceding PRINT command is an instruction to produce a report, in the output file, of the member end forces.

| LOAD LIST 1 4 5 |

A LOAD LIST command is a means of instructing the program to use only the listed load cases for further processing.

| PARAMETER |
| CODE AISC UNIFIED |
| BEAM 1.0 ALL |
| UNT 6.0 ALL |
| UNB 6.0 ALL |
| KY 0.5 ALL |
The PARAMETER command is used to specify the steel design parameters (information on these parameters can be obtained from the manual where the implementation of the code is explained). Design will be done according to the specifications of the AISC ASD Code. The BEAM parameter is specified to perform design at every 1/12th point along the member length. UNT and UNB represent the unsupported length of the flanges to be used for calculation of allowable bending stress. KY 0.5 ALL sets the effective length factor for column buckling about the local Y-axis to be 0.5 for ALL members.

CHECK CODE ALL

The above command instructs the program to perform a check to determine how you defined member sizes along with the latest analysis results meet the code requirements.

FINISH

This command terminates the STAAD run.

Input File

STAAD PLANE A PLANE FRAME STRUCTURE WITH TENSION BRACING
UNIT INCH KIP
SET NL 3
JOINT COORDINATES
1 0 0 0 3 480. 0 0
4 0 180. 0 6 480. 180. 0
7 240. 360. 0 ; 8 480. 360. 0
MEMBER INCIDENCE
1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
13 6 7 ;14 5 8
MEMBER TRUSS
9 TO 14
MEMBER PROP AMERICAN
1 TO 5 TABLE ST W12X26
6 7 8 TA ST W18X35
9 TO 14 TA LD L50505
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 2 3 PINNED
INACTIVE MEMBERS 9 TO 14
UNIT FT
LOADING 1 DEAD AND LIVE LOAD
MEMBER LOAD
6 8 UNI GY -1.0
7 UNI GY -1.5
PERFORM ANALYSIS
CHANGES
INACTIVE MEMBERS 10 11 13
LOADING 2 WIND FROM LEFT
JOINT LOAD
4 FX 30 ; 7 FX 15
PERFORM ANALYSIS
CHANGE
INACTIVE MEMBERS 9 12 14
LOADING 3 WIND FROM RIGHT
JOINT LOAD
6 FX -30 ; 8 FX -15
LOAD COMBINATION 4
1 0.75 2 0.75
LOAD COMBINATION 5
1 0.75 3 0.75
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 4 5
PARAMETER
CODE AISC UNIFIED
BEAM 1.0 ALL
UNT 6.0 ALL
UNB 6.0 ALL
KY 0.5 ALL
CHECK CODE ALL
FINISH

STAAD Output File

*****************************************************************************
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 22:57:10 *
* *
* Licensed to: Bentley Systems Inc *
*****************************************************************************

1. STAAD PLANE A PLANE FRAME STRUCTURE WITH TENSION BRACING

INPUT FILE: US-4 Inactive Members in a Braced Frame.STD
2. UNIT INCH KIP
3. SET NL 3
4. JOINT COORDINATES
5. 1 0 0 0 3 480. 0 0
6. 4 0 180. 0 6 480. 180. 0
7. 7 240. 360. 0 ; 8 480. 360. 0
8. MEMBER INCIDENCE
9. 1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
10. 8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
11. 13 6 7 ; 14 5 8
12. MEMBER TRUSS
13. 9 TO 14
14. MEMBER PROP AMERICAN
15. 1 TO 5 TABLE ST W12X26
16. 6 7 8 TA ST W18X35
17. 9 TO 14 TA LD L50505
18. DEFINE MATERIAL START
19. ISOTROPIC STEEL
20. E 29000
21. POISSON 0.3
22. DENSITY 283E-006
23. ALPHA 6E-006
24. DAMP 0.03
25. TYPE STEEL
26. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
27. END DEFINE MATERIAL
28. CONSTANT
29. MATERIAL STEEL ALL
30. SUPPORT
31. 1 2 3 PINNED
32. INACTIVE MEMBERS 9 TO 14
33. UNIT FT
34. LOADING 1 DEAD AND LIVE LOAD
35. MEMBER LOAD
36. 6 8 UNI GY -1.0
37. 7 UNI GY -1.5
38. PERFORM ANALYSIS
   A PLANE FRAME STRUCTURE WITH TENSION BRACING

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          8  NUMBER OF MEMBERS      14
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
39. CHANGES
40. INACTIVE MEMBERS 10 11 13
41. LOADING 2 WIND FROM LEFT
42. JOINT LOAD
43. 4 FX 30 ; 7 FX 15
44. PERFORM ANALYSIS
45. CHANGE
46. INACTIVE MEMBERS 9 12 14
47. LOADING 3 WIND FROM RIGHT
48. JOINT LOAD
49. 6 FX -30 ; 8 FX -15
50. LOAD COMBINATION 4
51. 1 0.75 2 0.75
52. LOAD COMBINATION 5
53. 1 0.75 3 0.75
54. PERFORM ANALYSIS
55. CHANGE
56. LOAD LIST ALL
57. PRINT MEMBER FORCES
MEMBER FORCES
   A PLANE FRAME STRUCTURE WITH TENSION BRACING

MEMBER END FORCES    STRUCTURE TYPE = PLANE
-----------------
ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
1 1 1 8.26 -0.67 0.00 0.00 0.00 -0.00
## Application Examples

EX. American Design Examples

---

### All Units Are -- KIP  FEET  (LOCAL)

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>6</td>
<td>9.86</td>
<td>2.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>15.84</td>
</tr>
<tr>
<td>8</td>
<td>-9.86</td>
<td>-2.20</td>
<td>0.00</td>
<td>0.00</td>
<td>17.11</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>10.95</td>
<td>0.37</td>
<td>0.00</td>
<td>0.00</td>
<td>2.68</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>10.95</td>
<td>0.37</td>
<td>0.00</td>
<td>0.00</td>
<td>2.82</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>-0.35</td>
<td>0.33</td>
<td>0.00</td>
<td>0.00</td>
<td>2.18</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.35</td>
<td>0.33</td>
<td>0.00</td>
<td>0.00</td>
<td>-2.78</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>6</td>
<td>15.61</td>
<td>1.92</td>
<td>0.00</td>
<td>0.00</td>
<td>13.89</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>15.61</td>
<td>1.92</td>
<td>0.00</td>
<td>0.00</td>
<td>14.95</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>7.14</td>
<td>1.40</td>
<td>0.00</td>
<td>0.00</td>
<td>10.30</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-7.14</td>
<td>-1.40</td>
<td>0.00</td>
<td>0.00</td>
<td>10.75</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>4</td>
<td>0.67</td>
<td>8.26</td>
<td>0.00</td>
<td>0.00</td>
<td>10.86</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-0.67</td>
<td>11.74</td>
<td>0.00</td>
<td>0.00</td>
<td>-44.76</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>29.78</td>
<td>-0.31</td>
<td>0.00</td>
<td>0.00</td>
<td>-3.25</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-29.78</td>
<td>0.31</td>
<td>0.00</td>
<td>0.00</td>
<td>-2.92</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
# Application Examples

EX. American Design Examples

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
<td>-33.43</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>33.43</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-25.87</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>25.87</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>-25.73</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>25.73</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>-19.29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>19.29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>-29.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>29.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

A PLANE FRAME STRUCTURE WITH TENSION BRACING

**MEMBER END FORCES**  STRUCTURE TYPE = PLANE

**ALL UNITS ARE -- KIP FEET (LOCAL)**
<table>
<thead>
<tr>
<th></th>
<th></th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>6</td>
<td>-17.56</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>6</td>
<td>17.56</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>-13.17</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>-17.77</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>17.77</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>-13.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>13.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

58. LOAD LIST 1 4 5
59. PARAMETER
60. CODE AISC UNIFIED
61. BEAM 1.0 ALL
62. UNT 6.0 ALL
63. UNB 6.0 ALL
64. KY 0.5 ALL
65. CHECK CODE ALL

STEEL DESIGN

A PLANE FRAME STRUCTURE WITH TENSION BRACING -- PAGE NO. 7

****************************************************

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

***NOTE : AISC 360-16 Design Statement for STAAD.Pro.

*** AXIS CONVENTION ***:

The capacity results and intermediate results in the report follow the notations and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
design results are reported in AISC 360-16 code axis convention.

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

SECTION FORCES AXIS MAPPING:

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
<tr>
<td>Tz</td>
<td>MX</td>
<td>Torsional moment.</td>
</tr>
<tr>
<td>My</td>
<td>MY</td>
<td>Bending moment about minor axis.</td>
</tr>
<tr>
<td>Mx</td>
<td>MZ</td>
<td>Bending moment about major axis.</td>
</tr>
</tbody>
</table>

*** DESIGN MESSAGES ***:

1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being \( \phi \) about \( \phi \) the relevant axis (X/Y), while the results for shear are reported as being for shear forces \( \phi \) along \( \phi \) the axis. E.g : Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

- F-T-B = Flexural-Torsional Buckling
- L-T-B = Lateral-Torsional Buckling
- F-L-B = Flange Local Buckling
- W-L-B = Web Local Buckling
- L-L-B = Leg Local Buckling
- C-F-Y = Compression Flange Yielding
- T-F-Y = Tension Flange Yielding

A PLANE FRAME STRUCTURE WITH TENSION BRACING  -- PAGE NO. 8

STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.1)

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).

- Member : 1

<table>
<thead>
<tr>
<th>Member No:</th>
<th>1</th>
<th>Profile: ST W12X26 (AISC SECTIONS)</th>
<th>Status:</th>
<th>PASS</th>
<th>Ratio:</th>
<th>0.137</th>
<th>Loadcase:</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>15.00</td>
<td>Ref: Eq.H1-3a(H1-1b)</td>
<td>Pz:</td>
<td>18.06</td>
<td>C</td>
<td>Vy:</td>
<td>-.6472</td>
<td>Vx:</td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My:</td>
<td>0.000</td>
<td>Mx:</td>
<td>9.708</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- Member : 2

<table>
<thead>
<tr>
<th>Member No:</th>
<th>2</th>
<th>Profile: ST W12X26 (AISC SECTIONS)</th>
<th>Status:</th>
<th>PASS</th>
<th>Ratio:</th>
<th>0.319</th>
<th>Loadcase:</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>15.00</td>
<td>Ref: Eq.H1-1a</td>
<td>Pz:</td>
<td>50.46</td>
<td>C</td>
<td>Vy:</td>
<td>-.1491</td>
<td>Vx:</td>
</tr>
<tr>
<td>Tz:</td>
<td>0.000</td>
<td>My:</td>
<td>0.000</td>
<td>Mx:</td>
<td>2.237</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- Member : 3
<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile:</th>
<th>Status:</th>
<th>Ratio:</th>
<th>Loadcase:</th>
<th>Location:</th>
<th>Ref:</th>
<th>Pz:</th>
<th>Vy:</th>
<th>Vx:</th>
<th>Mx:</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>ST W12X26</td>
<td>PASS</td>
<td>0.220</td>
<td>1</td>
<td>15.00</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>10.14</td>
<td>-2.197</td>
<td>0.000</td>
<td>19.84</td>
</tr>
<tr>
<td>4</td>
<td>ST W12X26</td>
<td>PASS</td>
<td>0.254</td>
<td>4</td>
<td>15.00</td>
<td>Eq.H1-3b</td>
<td>35.92</td>
<td>0.5957</td>
<td>0.000</td>
<td>-8.935</td>
</tr>
<tr>
<td>5</td>
<td>ST W12X26</td>
<td>PASS</td>
<td>0.192</td>
<td>1</td>
<td>15.00</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>9.864</td>
<td>2.197</td>
<td>0.000</td>
<td>-17.11</td>
</tr>
<tr>
<td>6</td>
<td>ST W18X35</td>
<td>PASS</td>
<td>0.268</td>
<td>4</td>
<td>20.00</td>
<td>Eq.H1-1b</td>
<td>22.84</td>
<td>-9.033</td>
<td>0.000</td>
<td>35.76</td>
</tr>
<tr>
<td>7</td>
<td>ST W18X35</td>
<td>PASS</td>
<td>0.327</td>
<td>1</td>
<td>0.00</td>
<td>Cl.F2.1</td>
<td>1.474</td>
<td>16.60</td>
<td>0.000</td>
<td>58.64</td>
</tr>
<tr>
<td>8</td>
<td>ST W18X35</td>
<td>PASS</td>
<td>0.179</td>
<td>1</td>
<td>10.00</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>2.197</td>
<td>0.1365</td>
<td>0.000</td>
<td>-31.52</td>
</tr>
<tr>
<td>9</td>
<td>LD L50505</td>
<td>PASS</td>
<td>0.126</td>
<td>4</td>
<td>0.00</td>
<td>Cl.D2</td>
<td>25.07</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>Member</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>------------------</td>
<td>------------------</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Member No:       10       Profile:  LD  L50505              (AISC SECTIONS)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Status:        PASS       Ratio:         0.097       Loadcase:        5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Location:       0.00       Ref:      Cl.D2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pz:           19.29     T     Vy:        0.000           Vx:      0.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tz:            0.000           My:        0.000           Mx:      0.000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| Member |  
|------------------|------------------|
| Member No:       11       Profile:  LD  L50505              (AISC SECTIONS) |
| Status:        PASS       Ratio:         0.113       Loadcase:        5   |
| Location:       0.00       Ref:      Cl.D2                                  |
| Pz:           22.42     T     Vy:        0.000           Vx:      0.000        |
| Tz:            0.000           My:        0.000           Mx:      0.000        |

| Member |  
|------------------|------------------|
| Member No:       12       Profile:  LD  L50505              (AISC SECTIONS) |
| Status:        PASS       Ratio:         0.084       Loadcase:        4   |
| Location:       0.00       Ref:      Cl.D2                                  |
| Pz:           16.69     T     Vy:        0.000           Vx:      0.000        |
| Tz:            0.000           My:        0.000           Mx:      0.000        |

| Member |  
|------------------|------------------|
| Member No:       13       Profile:  LD  L50505              (AISC SECTIONS) |
| Status:        PASS       Ratio:         0.066       Loadcase:        5   |
| Location:       0.00       Ref:      Cl.D2                                  |
| Pz:           13.17     T     Vy:        0.000           Vx:      0.000        |
| Tz:            0.000           My:        0.000           Mx:      0.000        |

| Member |  
|------------------|------------------|
| Member No:       14       Profile:  LD  L50505              (AISC SECTIONS) |
| Status:        PASS       Ratio:         0.067       Loadcase:        4   |
| Location:       0.00       Ref:      Cl.D2                                  |
| Pz:           13.33     T     Vy:        0.000           Vx:      0.000        |
| Tz:            0.000           My:        0.000           Mx:      0.000        |

66. FINISH

*********** END OF THE STAAD.Pro RUN ***********

**** DATE= APR 14,2019 TIME= 22:57:11 ****

* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/ *
* * Details about additional assistance from *
* Bentley and Partners can be found at program menu *
* Help->Technical Support *
* * Copyright (c) 1997-2017 Bentley Systems, Inc. *
* http://www.bentley.com *

### Related Links
- [Member Specification dialog](#) (on page 2962)
EX. US-5 Support Settlement on a Portal Frame

This example demonstrates the application of support displacement load (also known as a sinking support) on a space frame structure.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-5 Support Settlement on a Portal Frame.STD when you install the program.

Where:

\[ L_1 = 10 \text{ ft} \]
\[ L_2 = 20 \text{ ft} \]

Actual input is shown in bold lettering followed by explanation.

### STAAD SPACE TEST FOR SUPPORT DISPLACEMENT

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

### UNITS KIP FEET

Defines the input units for the data that follows.

### JOINT COORDINATES

<table>
<thead>
<tr>
<th>Joint</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>0.0</td>
<td>10.0</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>20.0</td>
<td>10.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>20.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>20.0</td>
<td>10.0</td>
<td>20.0</td>
</tr>
<tr>
<td>6</td>
<td>20.0</td>
<td>0.0</td>
<td>20.0</td>
</tr>
</tbody>
</table>

Joint number followed by X, Y and Z coordinates are provided above.
Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 3</td>
</tr>
<tr>
<td>4 3 5 ; 5 5 6</td>
</tr>
</tbody>
</table>

Defines the members by the joints to which they are connected.

<table>
<thead>
<tr>
<th>UNIT INCH MEMB PROP</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 5 PRIS AX 10. IZ 300. IY 300. IX 10.</td>
</tr>
</tbody>
</table>

Member properties have been defined above using the PRISMATIC attribute. Values of AX (area), IZ (moment of inertia about major axis), IY (moment of inertia about minor axis) and IX (torsional constant) are provided in INCH unit.

```
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 29000.
  POISSON 0.3
  DENSITY 283e-006
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

Material are defined using the DEFINE MATERIAL command and then assigned using the CONSTANT command.

<table>
<thead>
<tr>
<th>SUPPORT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 4 6 FIXED</td>
</tr>
</tbody>
</table>

Joints 1, 4 and 6 are fixed supports.

<table>
<thead>
<tr>
<th>LOADING 1 SINKING SUPPORT</th>
</tr>
</thead>
</table>

Load case 1 is initiated followed by a title.

<table>
<thead>
<tr>
<th>SUPPORT DISPLACEMENT LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 FY -0.50</td>
</tr>
</tbody>
</table>

Load 1 is a support displacement load which is also commonly known as a sinking support. FY signifies that the support settlement is in the global Y direction and the value of this settlement is 0.5 inch downward.

<table>
<thead>
<tr>
<th>PERFORM ANALYSIS</th>
</tr>
</thead>
</table>

This command instructs the program to proceed with the analysis.

<table>
<thead>
<tr>
<th>PRINT ANALYSIS RESULTS</th>
</tr>
</thead>
</table>

The above PRINT command instructs the program to print joint displacements, support reactions and member forces.

<table>
<thead>
<tr>
<th>FINISH</th>
</tr>
</thead>
</table>

This command terminates the STAAD run.

Input File

```
STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
UNITS KIP FEET
```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 10.0 0.0
3 20.0 10.0 0.0 ; 4 20.0 0.0 0.0
5 20. 10. 20. ; 6 20. 0. 20.
MEMBER INCIDENCE
1 1 2 3
4 3 5 ; 5 5 6
UNIT INCH
MEMB PROP
1 TO 5 PRIS AX 10. IZ 300. IY 300. IX 10.
DEFINE MATERIAL
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 4 6 FIXED
LOADING 1 SINKING SUPPORT
SUPPORT DISPLACEMENT LOAD
4 FY -0.50
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH

STAAD Output File

*************
* STAAD.Pro CONNECT Edition
* Version 22.01.00.**
* Proprietary Program of
* Bentley Systems, Inc.
* Date= APR 14, 2019
* Time= 22:57:15
*
* Licensed to: Bentley Systems Inc
*
*****************************************************************************
1. STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
2. UNITS KIP FEET
3. JOINT COORDINATES
4. 1 0.0 0.0 0.0 ; 2 0.0 10.0 0.0
5. 3 20.0 10.0 0.0 ; 4 20.0 0.0 0.0
6. 5 20. 10. 20. ; 6 20. 0. 20.
7. MEMBER INCIDENCE
8. 1 1 2 3
9. 4 3 5 ; 5 5 6
10. UNIT INCH
11. MEMB PROP
12. 1 TO 5 PRIS AX 10. IZ 300. IY 300. IX 10.
13. DEFINE MATERIAL START
14. ISOTROPIC STEEL
15. E 29000.
16. POISSON 0.3
17. DENSITY 283E-006
18. ALPHA 6E-006
19. DAMP 0.03
20. TYPE STEEL
21. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
22. END DEFINE MATERIAL
23. CONSTANT
24. MATERIAL STEEL ALL
25. SUPPORT
26. 1 4 6 FIXED
27. LOADING 1 SINKING SUPPORT
28. SUPPORT DISPLACEMENT LOAD
29. 4 FY -0.50
30. PERFORM ANALYSIS
31. PRINT ANALYSIS RESULTS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

ANALYSIS RESULTS
TEST FOR SUPPORT DISPLACEMENT
-----------------------------------
JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = SPACE
------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
2    1    0.09125  -0.00040  -0.01078  -0.00014   0.00050  -0.00154
3    1    0.09118  -0.49919  -0.09118  -0.00154   0.00000  -0.00154
4    1    0.00000  -0.50000   0.00000   0.00000   0.00000   0.00000
5    1    0.01078  -0.00040  -0.09125  -0.00154  -0.00050  -0.00014
6    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000

SUPPORT REACTIONS -UNIT KIP INCH  STRUCTURE TYPE = SPACE
-----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z   MOM-X   MOM-Y   MOM Z
1    1      0.08      0.97      0.15     19.22    -0.46    107.07
4    1      0.07     -1.95     -0.07    107.18     0.00    107.18
6    1     -0.15      0.97     -0.08    107.07     0.46    19.22

MEMBER END FORCES  STRUCTURE TYPE = SPACE
-----------------------------------
ALL UNITS ARE -- KIP INCH (LOCAL)
MEMBER  LOAD  JT  AXIAL  SHEAR-Y  SHEAR-Z  TORSION  MOM-Y  MOM-Z
1 1 1 0.97  -0.08  0.15  -0.46  -19.22  107.07
2 1 2 0.08   0.97  0.15   0.65  -0.46  116.64
3 1 3 0.08  -0.97  -0.15  -0.65  -36.66  116.82
4 1 4 0.08  -0.08  0.07  -0.00  -116.17  116.17
5 1 5 0.08  -0.08  0.07  -0.00  -116.17  116.17
6 1 6 0.08  -0.08  0.07  -0.00  -116.17  116.17

STAAD.Pro 4406 User Manual
EX. US-6 Prestress and Poststress Loading

This is an example of prestress loading in a plane frame structure.

It covers two situations:

1. From the member on which it is applied, the prestressing effect is transmitted to the rest of the structure through the connecting members (known in the program as PRESTRESS load).

2. The prestressing effect is experienced by the member(s) alone and not transmitted to the rest of the structure (known in the program as POSTSTRESS load).

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-6 Prestress and Poststress Loading.STD when you install the program.
Figure 459: Example Problem No. 6

Where:

L1 = 15 ft
L2 = 20 ft
L3 = 40 ft

Actual input is shown in bold lettering followed by explanation.

STAAD PLANE FRAME WITH PRESTRESSING LOAD

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

UNIT KIP FT

Defines the input units for the data that follows.

JOINT COORD
1  0.  0.  ;  2  40.  0.  ;  3  0.  20.  ;  4  40.  20.  
5  0.  35.  ;  6  40.  35.  ;  7  0.  50.  ;  8  40.  50.  

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCE
1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
Defines the members by the joints to which they are connected.

```
SUPPORT
  1  2  FIXED
```

The supports at joints 1 and 2 are defined to be fixed supports.

```
MEMB PROP
  1  TO  9  PRI  AX  2.2  IZ  1.0
```

Member properties are provided using the PRI (prismatic) attribute. Values of area (AX) and moment of inertia about the major axis (IZ) are provided.

```
UNIT INCH
DEFINE MATERIAL START
  ISOTROPIC  CONCRETE
  E  3000.
  POISSON  0.17
  DENSITY  8.7e-005
  ALPHA  5e-006
  DAMP  0.05
  G  1346.15
  TYPE  CONCRETE
  STRENGTH  FCU  4
END DEFINE MATERIAL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` command is used to assign the material to all members. Length unit is changed from FT to INCH to facilitate the input.

```
LOADING 1  PRESTRESSING LOAD
MEMBER  PRESTRESS
  7  8  FORCE  300.  ES  3.  EM -12.  EE  3.
```

Load case 1 is initiated along with an accompanying title. Load 1 contains PRESTRESS load. Members 7 and 8 have a cable force of 300 kips. The location of the cable at the start (ES) and end (EE) is 3 inches above the center of gravity while at the middle (EM) it is 12 inches below the c.g. The assumptions and facts associated with this type of loading are explained in section 1 of the Technical Reference Manual.

```
LOADING 2  POSTSTRESSING LOAD
MEMBER  POSTSTRESS
  7  8  FORCE  300.  ES  3.  EM -12.  EE  3.
```

Load case 2 is initiated along with an accompanying title. Load 2 is a POSTSTRESS load. Members 7 and 8 have cable force of 300 kips. The location of the cable is the same as in load case 1. For a difference between PRESTRESS loading and POSTSTRESS loading, as well as additional information about both types of loads, please refer to section 1 of the Technical Reference Manual.

```
PERFORM  ANALYSIS
```

This command instructs the program to perform the analysis.

```
UNIT  FT
PRINT  ANALYSIS  RESULT
```

The preceding command is an instruction to write joint displacements, support reactions and member forces in the output file. The preceding line causes the results to be written in the length unit of feet.

```
FINISH
```

This command terminates the STAAD run.
Input File

STAAD PLANE  FRAME WITH PRESTRESSING LOAD
UNIT KIP FT
JOINT COORD
1  0.  0. ; 2  40.  0. ; 3  0.  20. ; 4  40.  20. ;
5  0.  35. ; 6  40.  35. ; 7  0.  50. ; 8  40.  50.
MEMBER INCIDENCE
1  1  3 ; 2  3  5 ; 3  5  7 ; 4  2  4 ; 5  4  6 ;
6  6  8 ; 7  3  4 ; 8  5  6 ; 9  7  8
SUPPORT
1  2  FIXED
MEMB PROP
1  TO 9  PRI AX 2.2  IZ 1.0
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3000.
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
LOADING 1 PRESTRESSING LOAD
MEMBER PRESTRESS
7  8  FORCE 300.  ES 3.  EM -12.  EE 3.
LOADING 2 POSTSTRESSING LOAD
MEMBER POSTSTRESS
7  8  FORCE 300.  ES 3.  EM -12.  EE 3.
PERFORM ANALYSIS
UNIT FT
PRINT ANALYSIS RESULT
FINISH

STAAD Output File
4. 1 0. 0. ; 2 40. 0. ; 3 0. 20. ; 4 40. 20.
5. 5 0. 35. ; 6 40. 35. ; 7 0. 50. ; 8 40. 50.
6. MEMBER INCIDENCE
7. 1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
8. 6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
9. SUPPORT
10. 1 2 FIXED
11. MEMB PROP
12. 1 TO 9 PRI AX 2.2 IZ 1.0
13. UNIT INCH
14. DEFINE MATERIAL START
15. ISOTROPIC CONCRETE
17. POISSON 0.17
18. DENSITY 8.7E-005
19. ALPHA 5E-006
20. DAMP 0.05
21. G 1346.15
22. TYPE CONCRETE
23. STRENGTH FCU 4
24. END DEFINE MATERIAL
25. CONSTANTS
26. MATERIAL CONCRETE ALL
27. LOADING 1 PRESTRESSING LOAD
28. MEMBER PRESTRESS
29. 7 8 FORCE 300. ES 3. EM -12. EE 3.
30. LOADING 2 POSTSTRESSING LOAD
31. MEMBER POSTSTRESS
32. 7 8 FORCE 300. ES 3. EM -12. EE 3.
33. PERFORM ANALYSIS
   FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.  2
   PROBLEM STATISTICS
   -----------------------------------
   NUMBER OF JOINTS          8  NUMBER OF MEMBERS       9
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
   Using 64-bit analysis engine.
   SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      18
   TOTAL LOAD COMBINATION CASES =     0  SO FAR.
34. UNIT FT
35. PRINT ANALYSIS RESULT
ANALYSIS RESULT
   FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.  3
   JOINT DISPLACEMENT (INCH RADIANS) STRUCTURE TYPE = PLANE
   -----------------------------------
   JOINT  LOAD  X-TRANS  Y-TRANS  Z-TRANS  X-ROTAN  Y-ROTAN  Z-ROTAN
   1  1  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   2  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   3  1  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   2  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   4  1  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   2  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   5  1  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   2  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   6  1  0.00000  0.00000  0.00000  0.00000  0.00000  0.00000
   0.00000 -0.00000 -0.00000 -0.00000
### Frame with Prestressing Load

**Support Reactions**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-6.71</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>58.62</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>6.71</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-58.62</td>
<td></td>
</tr>
</tbody>
</table>

### Member End Forces

**All Units Are -- KIP Feet**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>6.71</td>
<td>0.00</td>
<td>0.00</td>
<td>58.62</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.00</td>
<td>-6.71</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>75.67</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

### Frame with Prestressing Load

**Member End Forces**

**All Units Are -- KIP Feet**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>7</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>
EX. US-7 Modeling Offset Connections in a Frame

This example illustrates modeling of structures with offset connections. Offset connections arise when the center lines of the connected members do not intersect at the connection point. The connection eccentricity behaves as a rigid link and is modeled through specification of MEMBER OFFSETS.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-7 Modeling Offset Connections in a Frame.STD when you install the program.

![Figure 460: Example Problem No. 7](image-url)
Where:

- \( L_1 = 15 \text{ ft} \)
- \( L_2 = 20 \text{ ft} \)
- \( L_3 = 7 \text{ in} \)
- \( L_4 = 6 \text{ in} \)

Actual input is shown in bold lettering followed by explanation.

**STAAD PLANE TEST FOR MEMBER OFFSETS**

Every input has to start with the term **STAAD**. The term **PLANE** signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

**UNIT FT KIP**

Defines the input units for the data that follows.

**JOINT COORD**

1 0. 0.; 2 20. 0.; 3 0. 15.
4 20. 15.; 5 0. 30.; 6 20. 30.

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

**SUPPORT**

1 2 PINNED

Pinned supports are specified at joints 1 and 2. The word **PINNED** signifies that no moments will be carried by these supports.

**MEMB INCI**

1 1 3 2; 3 3 5 4
5 3 4; 6 5 6; 7 1 4

Defines the members by the joints to which they are connected.

**MEMB PROP AMERICAN**

1 TO 4 TABLE ST W14X90
5 6 TA ST W12X26
7 TA LD L80608

Member properties are assigned from the American (AISC) steel table for all members. The word **ST** stands for standard single section. **LD** stands for long leg back-to-back double angle.

**Note:** This problem uses a steel size from an older AISC catalog. In order to use this shape for analysis and design, you must select **AISCSections** or **aiscsections_all_editions** database for use as the **American** section profile table.

**UNIT INCH**

**MEMB OFFSET**

5 6 START 7.0 0.0 0.0
5 6 END -7.0 0.0 0.0
7 END -7.0 -6.0 0.0
The preceding specification states that an OFFSET is located at the START/END joint of the members. The X, Y, and Z global coordinates of the offset distance from the corresponding incident joint are also provided. These attributes are applied to members 5, 6, and 7.

```
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 29000.
  POISSON 0.3
  DENSITY 283e-006
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
LOADING 1 WIND LOAD
```

Load case 1 is initiated followed by a title.

```
JOINT LOAD
  3 FX 50.; 5 FX 25.0
```

Load 1 contains joint loads at nodes 3 and 5. FX indicates that the load is a force in the global X direction.

```
PERFORM ANALYSIS
```

The above command is an instruction to perform the analysis.

```
UNIT FT
PRINT FORCES
PRINT REACTIONS
```

The above PRINT commands are instructions for writing the member forces and support reactions to the output file. The preceding line causes the results to be written in the length unit of feet.

```
FINISH
```

This command terminates a STAAD run.

```
Input File
```

```
STAAD PLANE TEST FOR MEMBER OFFSETS
UNIT FT KIP
JOINT COORD
  1 0. 0.; 2 20. 0.; 3 0. 15.
  4 20. 15.; 5 0. 30.; 6 20. 30.
MEMB INCI
  1 1 3 2; 3 3 5 4
  5 3 4; 6 5 6; 7 1 4
MEMB PROP AMERICAN
  1 TO 4 TABLE ST W14X90
  5 6 TA ST W12X26
  7 TA LD L80608
UNIT INCH
MEMB OFFSET
  5 6 START 7.0 0.0 0.0
  5 6 END -7.0 0.0 0.0
```
DEFINE MATERIAL
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL

CONSTANT
MATERIAL STEEL ALL

SUPPORT
1 2 PINNED

LOADING 1 WIND LOAD
JOINT LOAD
3 FX 50. ; 5 FX 25.0

PERFORM ANALYSIS
UNIT FT
PRINT FORCES
PRINT REACTIONS
FINISH

STAAD Output File

1. STAAD PLANE TEST FOR MEMBER OFFSETS
INPUT FILE: US-7 Modeling Offset Connections in a Frame.STD

2. UNIT FT KIP
3. JOINT COORD
4. 1 0. 0. ; 2 20. 0. ; 3 0. 15.
5. 4 20. 15. ; 5 0. 30. ; 6 20. 30.
6. MEMB INCI
7. 1 1 3 2; 3 5 4
8. 5 4 6 ; 6 5 6 ; 7 1 4
9. MEMB PROP AMERICAN
10. 1 TO 4 TABLE ST W14X90
11. 5 6 TA ST W12X26
12. 7 TA LD L80608
13. UNIT INCH
14. MEMB OFFSET
15. 5 6 START 7.0 0.0 0.0
16. 5 6 END 7.0 -6.0 0.0
17. 7 END -7.0 -6.0 0.0
18. DEFINE MATERIAL START
19. ISOTROPIC STEEL
20. E 29000
21. POISSON 0.3
22. DENSITY 283E-006
23. ALPHA 6E-006
24. DAMP 0.03
25. TYPE STEEL
26. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
27. END DEFINE MATERIAL
28. CONSTANT
29. MATERIAL STEEL ALL
30. SUPPORT
31. 1 2 PINNED
32. LOADING 1 WIND LOAD
33. JOINT LOAD
34. 3 FX 50 ; 5 FX 25.0
35. PERFORM ANALYSIS
   TEST FOR MEMBER OFFSETS  -- PAGE NO.  2

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS       7
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
36. UNIT FT
37. PRINT FORCES

FORCES
   TEST FOR MEMBER OFFSETS  -- PAGE NO.  3
MEMBER END FORCES    STRUCTURE TYPE = PLANE
-----------------
ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER LOAD  JT   AXIAL  SHEAR-Y  SHEAR-Z  TORSION  MOM-Y   MOM-Z
1    1    1   -10.69   -4.61   0.00     0.00     0.00      3.56
3    1    2     75.00    5.73   0.00     0.00     0.00     -85.95
2    1    3    -6.73   11.92   0.00     0.00     0.00    112.47
4    1    4     6.73   -11.92   0.00     0.00     0.00     66.37
5    1    3    -66.53    3.96   0.00     0.00     0.00    -37.49
6    1    5    -66.53    3.96   0.00     0.00     0.00    -37.16
7    1    1   -106.85   -0.46   0.00     0.00     0.00      -3.56
4    1    5    106.85     0.46   0.00     0.00     0.00      -7.64

*************** END OF LATEST ANALYSIS RESULT ***************

38. PRINT REACTIONS
REACTION
   TEST FOR MEMBER OFFSETS  -- PAGE NO.  4
SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
-----------------
JOINT LOAD   FORCE-X  FORCE-Y  FORCE-Z  MOM-X  MOM-Y  MOM Z
1    1  -80.73   -75.00     0.00     0.00     0.00
2    1     5.73    75.00     0.00     0.00     0.00

*************** END OF LATEST ANALYSIS RESULT ***************
EX. US-8 Concrete Design for a Space Frame

In this example, concrete design is performed on some members of a space frame structure. Design calculations consist of computation of reinforcement for beams and columns. Secondary moments on the columns are obtained through the means of a P-Delta analysis.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-8 Concrete Design for a Space Frame.STD when you install the program.
The above example represents a space frame, and the members are made of concrete. The input in the next page will show the dimensions of the members.

Two load cases, namely one for dead plus live load and another with dead, live and wind load, are considered in the design.

Actual input is shown in bold lettering followed by explanation.

**STAAD SPACE FRAME WITH CONCRETE DESIGN**

Every input has to start with the term **STAAD**. The word **SPACE** signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

**UNIT KIP FT**

Defines the input units for the data that follows.

**JOINT COORDINATE**

1  0  0  0 ; 2  18  0  0 ; 3  38  0  0
4  0  0  24 ; 5  18  0  24 ; 6  38  0  24
7  0  12  0 ; 8  18  12  0 ; 9  38  12  0
10 0  12  24 ; 11  18  12  24 ; 12  38  12  24
13  18  24  0 ; 14  38  24  0 ; 15  18  24  24

Joint number followed by X, Y and Z coordinates are provided above.
Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCE

1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
9 6 12 ; 10 12 16 ; 11 7 8 12
13 10 11 14 ; 15 13 14 ; 16 15 16
17 7 10 ; 18 8 11 ; 19 9 12
20 13 15 ; 21 14 16

Defines the members by the joints to which they are connected.

UNIT INCH
MEMB PROP
1 2 PRISMATIC YD 12.0 IZ 509. IY 509. IX 1018.
3 TO 10 PR YD 12.0 ZD 12.0 IZ 864. IY 864. IX 1279.
11 TO 21 PR YD 21.0 ZD 16.0 IZ 5788. IY 2953. IX 6497.

All member properties are provided using the PRISMATIC option. YD and ZD stand for depth and width. If ZD is not provided, a circular shape with diameter = YD is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. For this particular example, moments of inertia (IZ, IY) and torsional constant (IX) are provided, so these will not be recalculated. The IX, IY, and IZ values provided in this example are only half the values of a full section to account for the fact that the full moments of inertia will not be effective due to cracking of concrete. Clause 10.11.1 of ACI 318-99 offers some guidelines on the amount of reduction to be applied on the gross section moment of inertia for beams, columns, walls and slabs to account for cracking.

DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FT

The DEFINE MATERIAL command is used to define material properties for concrete. The CONSTANT command is used to assign this to all members.

SUPPORT
1 TO 6 FIXED

Joints 1 to 6 are fixed supports.

LOAD 1 (1.4DL + 1.7LL)

Load case 1 is initiated followed by a title.

SELF Y -1.4

The selfweight of the structure is applied in the global Y direction with a -1.4 factor. Since global Y is vertically upward, the negative factor indicates that this load will act downwards.
Load 1 contains member loads also. \( Y \) indicates that the load is in the local \( Y \) direction. The word \textit{UNI} stands for uniformly distributed load.

| Load 2 \( 0.75 \times (1.4DL + 1.7LL + 1.7WL) \) |

Load case 2 is initiated followed by a title.

| \text{REPEAT LOAD} | 1 \( 0.75 \) |

The preceding command will gather the load data values from load case 1, multiply them with a factor of 0.75 and utilize the resulting values in load 2.

| \text{Joint Load} | 15 16 \( FZ \) 8.5 |
| 11 \( FZ \) 20.0 |
| 12 \( FZ \) 16.0 |
| 10 \( FZ \) 8.5 |

Load 2 contains some additional joint loads also. \( FZ \) indicates that the load is a force in the global \( Z \) direction.

| \text{PDELTA ANALYSIS} |

This command instructs the program to proceed with the analysis. The analysis type is \textit{P-DELTA} indicating that second-order effects are to be calculated.

| \text{Print Forces List} 2 5 9 14 16 |

Member end forces are printed using the above \textit{PRINT} commands. The \textit{LIST} option restricts the print output to the members listed.

| \text{Start Concrete Design} |

The above command initiates a concrete design.

| \text{Code ACI} |
| \text{Track 1.0 Memb 14} |
| \text{Track 2.0 Memb 16} |
| \text{MaxMain 11 All} |

The values for the concrete design parameters are defined in the above commands. Design is performed per the \textit{ACI 318 Code}. The \textit{Track} value dictates the extent of design related information that should appear in the output. \textit{MaxMain} indicates that the maximum size of main reinforcement is the \#11 bar. These parameters are described in the manual where American concrete design related information is available.

| \text{Design Beam} 14 16 |

The above command instructs the program to design beams 14 and 16 for flexure, shear, and torsion.

| \text{Design Column} 2 5 |

The above command instructs the program to design columns 2 and 5 for axial load and biaxial bending.

| \text{End Concrete Design} |

This will end the concrete design.

| \text{Finish} |

This command terminates the STAAD run.
Input File

STAAD SPACE FRAME WITH CONCRETE DESIGN
UNIT KIP FT
JOINT COORDINATE
1 0 0 0 ; 2 18 0 0 ; 3 38 0 0
4 0 0 24 ; 5 18 0 24 ; 6 38 0 24
7 0 12 0 ; 8 18 12 0 ; 9 38 12 0
10 0 12 24 ; 11 18 12 24 ; 12 38 12 24
13 18 24 0 ; 14 38 24 0 ; 15 18 24 24
16 38 24 24
MEMBER INCIDENCE
1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
9 6 12 ; 10 12 16 ; 11 7 8 12
13 10 11 14 ; 15 13 14 ; 16 15 16
17 7 10 ; 18 8 11 ; 19 9 12
20 13 15 ; 21 14 16
UNIT INCH
MEMB PROP
1 2 PRISMATIC YD 12.0 IZ 509. IY 509. IX 1018.
3 TO 10 PR YD 12.0 ZD 12.0 IZ 864. IY 864. IX 1279.
11 TO 21 PR YD 21.0 ZD 16.0 IZ 5788. IY 2953. IX 6497.
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FT
SUPPORT
1 TO 6 FIXED
LOAD 1 (1.4DL + 1.7LL)
SELF Y -1.4
MEMB LOAD
11 TO 16 UNI Y -2.8
11 TO 16 UNI Y -5.1
LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
REPEAT LOAD
1 0.75
JOINT LOAD
15 16 FZ 8.5
11 FZ 20.0
12 FZ 16.0
10 FZ 8.5
PDELTA ANALYSIS
PRINT FORCES LIST 2 5 9 14 16
START CONCRETE DESIGN
CODE ACI
TRACK 1.0 MEMB 14
1. STAAD SPACE FRAME WITH CONCRETE DESIGN

INPUT FILE: US-8 Concrete Design for a Space Frame.STD

2. UNIT KIP FT

3. JOINT COORDINATE

4. 1 0 0 0 ; 2 18 0 0 ; 3 38 0 0
5. 4 0 0 24 ; 5 18 0 24 ; 6 38 0 24
6. 7 0 12 0 ; 8 18 12 0 ; 9 38 12 0
7. 10 0 12 24 ; 11 18 12 24 ; 12 38 12 24
8. 13 18 24 0 ; 14 38 24 0 ; 15 18 24 24
9. 16 38 24 24

10. MEMBER INCIDENCE

11. 1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
12. 5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
13. 9 6 12 ; 10 12 16 ; 11 7 8 12
14. 13 10 11 14 ; 15 13 14 ; 16 15 16
15. 17 7 10 ; 18 8 11 ; 19 9 12
16. 20 13 15 ; 21 14 16
17. UNIT INCH

18. MEMB PROP

19. 1 2 PRISMATIC YD 12.0 IZ 509. IY 509. IX 1018.
20. 3 TO 10 PR YD 12.0 ZD 12.0 IZ 864. IY 864. IX 1279.
21. 11 TO 21 PR YD 21.0 ZD 16.0 IZ 5788. IY 2953. IX 6497.

22. DEFINE MATERIAL START

23. ISOTROPIC CONCRETE

24. Constants

25. MATERIAL CONCRETE ALL

26. STRENGTH FCU 4

27. END DEFINE MATERIAL

28. CONSTANTS

29. MATERIAL CONCRETE

30. SUPPORT

---

STAAD Output File
37. 1 TO 6 FIXED
38. LOAD 1 (1.4DL + 1.7LL)
   -- PAGE NO. 2
39. SELF Y -1.4
40. MEMB LOAD
41. 11 TO 16 UNI Y -2.8
42. 11 TO 16 UNI Y -5.1
43. LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
44. REPEAT LOAD
45. 1 0.75
46. JOINT LOAD
47. 15 16 FZ 8.5
48. 11 FZ 20.0
49. 12 FZ 16.0
50. 10 FZ 8.5
51. PDELTA ANALYSIS
   PROBLEM STATISTICS
   -----------------------------------
   NUMBER OF JOINTS         16  NUMBER OF MEMBERS      21
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
   Using 64-bit analysis engine.
   SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 60
   TOTAL LOAD COMBINATION CASES = 0  SO FAR.
   ++ Adjusting Displacements.
52. PRINT FORCES LIST 2 5 9 14 16
53. FORCES LIST 2 5
   \---------------
   \---------------
   \---------------
   \---------------
   ALL UNITS ARE -- KIP FEET (LOCAL )
   MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
2    1    4   68.01 -4.02 -0.69  0.00  2.85  -17.70
    10  -66.03  4.02  0.69 -0.00  5.48  -30.94
2    4   54.80 -3.35 -6.58  1.03 41.93  -15.37
    10  -53.31  3.35  6.58 -1.03 41.50  -25.37
5    1    5  289.52 -0.45 -0.72  0.00  3.08  -4.21
    11  -286.99  0.45  0.72 -0.00  5.54   -3.04
2    5   227.65 -0.88  12.16  0.90 88.74  -6.81
    11 -225.76  0.88  12.16 -0.90 82.17  -5.84
9    1    6   170.70  4.47 -0.67 -0.00  2.79   16.47
    12  -168.18  4.47  0.67  0.00  5.27   36.05
2    6   139.15  2.84 -13.36  0.17 92.01   8.79
    12 -137.26  2.84  13.36 -0.17 84.24  24.01
14   1    1   -9.12  97.25  0.00 -0.23 -0.01 371.86
    12   9.12  70.57 -0.00  0.23 -0.00 -104.95
2    11  -7.75  73.11  0.64 -0.87 -8.77 280.25
    12   7.75  52.76 -0.64  0.87 -3.99 -76.74
16   1    5   13.58  84.52  0.00  0.03 -0.00 104.14
    16  -13.58  83.30  0.00 -0.03  0.00 -91.95
2    15  10.19  63.36  0.07 -0.08 -0.79  78.03
    16 -10.19  62.50 -0.07  0.08 -0.59 -69.48
   ************** END OF LATEST ANALYSIS RESULT **************
54. CODE ACI
55. TRACK 1.0 MEMB 14
DESIGN SUMMARY

| Status         : | Pass     | Type    : | Beam     | Length: | 20.000 |
| Critical Ratio : | 0.940    | Criteria: | Flexure  |         |        |
| Critical Clause: | 9.5.2    |          |          |         |        |

CROSS SECTION

Shape: Rectangular | Width: | 1.33 | Depth: | 1.75 |

LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Nums</th>
<th>Size</th>
<th>Location</th>
<th>Distance</th>
<th>Anchor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right</td>
<td>5</td>
<td>#4</td>
<td>0.00</td>
<td>20.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Top</td>
<td>3</td>
<td>#8</td>
<td>0.00</td>
<td>20.00</td>
<td>Yes</td>
</tr>
<tr>
<td>Top</td>
<td>3</td>
<td>#8</td>
<td>0.00</td>
<td>8.28</td>
<td>Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>3</td>
<td>#5</td>
<td>0.00</td>
<td>20.00</td>
<td>No</td>
</tr>
<tr>
<td>Bottom</td>
<td>2</td>
<td>#5</td>
<td>2.68</td>
<td>17.32</td>
<td>No</td>
</tr>
<tr>
<td>Left</td>
<td>5</td>
<td>#4</td>
<td>0.00</td>
<td>20.00</td>
<td>Yes</td>
</tr>
</tbody>
</table>

TRANSVERSE BAR LAYOUT

Zone | Dir. | From | To | Asv | Reqd. | Prov. | Nums | Size | Spacing | Legs |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>10.00</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>47</td>
<td>#4</td>
<td>0.22</td>
<td>2</td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>10.00</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>47</td>
<td>#4</td>
<td>0.22</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Y</td>
<td>10.00</td>
<td>15.00</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>11</td>
<td>#4</td>
<td>0.50</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Z</td>
<td>10.00</td>
<td>15.00</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>11</td>
<td>#4</td>
<td>0.50</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>Y</td>
<td>15.00</td>
<td>20.00</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>20</td>
<td>#4</td>
<td>0.26</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>Z</td>
<td>15.00</td>
<td>20.00</td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>20</td>
<td>#4</td>
<td>0.26</td>
<td>2</td>
</tr>
</tbody>
</table>

Member: 14 Design Ends
### DESIGN INPUTS

| Concrete | $F_c$       | 576.000 |                     | $E_c$     | 0.519E+06 |
| Steel    | $F_y$ (main) | 8639.999 | $F_y$ (trans) 8639.999 | $E_s$     | 0.418E+07 |
| Cover    | Top         | 0.125   | Bottom        0.125 | Sides     | 0.125  |

### Design Messages

**WARNINGS:** DESIGN FOR MEMBER 16

1) Review shear min density/max spacing Cl 9.7.6.2.2

### CRITICAL STRENGTH RESULTS

<table>
<thead>
<tr>
<th>Category</th>
<th>Demand</th>
<th>Min Capacity</th>
<th>Max Capacity</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial</td>
<td>-13.584</td>
<td>-757.984</td>
<td>300.780</td>
<td>0.018</td>
</tr>
<tr>
<td>Flexure</td>
<td>214.510</td>
<td>-185.248</td>
<td>265.630</td>
<td>0.808</td>
</tr>
<tr>
<td>Shear Y</td>
<td>84.523</td>
<td>-123.550</td>
<td>123.550</td>
<td>0.684</td>
</tr>
<tr>
<td>Shear Z</td>
<td>-0.073</td>
<td>-23.476</td>
<td>23.476</td>
<td>0.003</td>
</tr>
<tr>
<td>Torsion</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

### LONGITUDINAL BAR DETAILS AT CROSS SECTIONS

<table>
<thead>
<tr>
<th>Distance</th>
<th>Position</th>
<th>Ast-reqd</th>
<th>Ast-prov</th>
<th>No(s)bars</th>
<th>Size</th>
<th>No of Layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>Top</td>
<td>0.007</td>
<td>0.008</td>
<td>6</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>0.010</td>
<td>0.016</td>
<td>3</td>
<td># 8</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Left</td>
<td>0.007</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Right</td>
<td>0.007</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td>5.000</td>
<td>Top</td>
<td>0.002</td>
<td>0.008</td>
<td>6</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>0.012</td>
<td>0.033</td>
<td>6</td>
<td># 8</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Left</td>
<td>0.007</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Right</td>
<td>0.007</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td>10.000</td>
<td>Top</td>
<td>0.002</td>
<td>0.008</td>
<td>6</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>0.030</td>
<td>0.033</td>
<td>6</td>
<td># 8</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Left</td>
<td>0.002</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Right</td>
<td>0.002</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td>15.000</td>
<td>Top</td>
<td>0.002</td>
<td>0.008</td>
<td>6</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>0.019</td>
<td>0.033</td>
<td>6</td>
<td># 8</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Left</td>
<td>0.002</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Right</td>
<td>0.002</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td>20.000</td>
<td>Top</td>
<td>0.007</td>
<td>0.008</td>
<td>6</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Bottom</td>
<td>0.010</td>
<td>0.016</td>
<td>3</td>
<td># 8</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Left</td>
<td>0.007</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Right</td>
<td>0.007</td>
<td>0.007</td>
<td>5</td>
<td># 4</td>
<td>1</td>
</tr>
</tbody>
</table>

### LONGITUDINAL BAR LAYOUT

<table>
<thead>
<tr>
<th>Position</th>
<th>Nums</th>
<th>Size</th>
<th>Location Start</th>
<th>Location End</th>
<th>Distance From Face</th>
<th>Anchor Start End</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right</td>
<td>5</td>
<td># 4</td>
<td>0.00</td>
<td>20.00</td>
<td>0.15</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Top</td>
<td>6</td>
<td># 4</td>
<td>0.00</td>
<td>20.00</td>
<td>0.15</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>3</td>
<td># 8</td>
<td>0.00</td>
<td>20.00</td>
<td>0.17</td>
<td>Yes Yes</td>
</tr>
<tr>
<td>Bottom</td>
<td>3</td>
<td># 8</td>
<td>4.06</td>
<td>18.77</td>
<td>0.17</td>
<td>No No</td>
</tr>
<tr>
<td>Left</td>
<td>5</td>
<td># 4</td>
<td>0.00</td>
<td>20.00</td>
<td>0.15</td>
<td>Yes Yes</td>
</tr>
</tbody>
</table>
## TRANSVERSE BAR LAYOUT

<table>
<thead>
<tr>
<th>Zone</th>
<th>Dir.</th>
<th>From</th>
<th>To</th>
<th>Reqd.</th>
<th>Prov.</th>
<th>Nums</th>
<th>Size</th>
<th>Spacing</th>
<th>Legs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y</td>
<td>0.00</td>
<td>10.00</td>
<td>0.00</td>
<td>0.00</td>
<td>56</td>
<td># 4</td>
<td>0.18</td>
<td>2</td>
</tr>
<tr>
<td>1</td>
<td>Z</td>
<td>0.00</td>
<td>10.00</td>
<td>0.00</td>
<td>0.00</td>
<td>56</td>
<td># 4</td>
<td>0.18</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Y</td>
<td>10.00</td>
<td>20.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38</td>
<td># 4</td>
<td>0.27</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Z</td>
<td>10.00</td>
<td>20.00</td>
<td>0.00</td>
<td>0.00</td>
<td>38</td>
<td># 4</td>
<td>0.27</td>
<td>2</td>
</tr>
</tbody>
</table>

---

### Member: 16 Design Ends

59. DESIGN COLUMN 2 5

FRAME WITH CONCRETE DESIGN

FRAME WITH CONCRETE DESIGN

**STAAD.PRO CONCRETE DESIGN - (ACI-318-14)** v2.0

---

**Units: KIP, FEET** (Unless Noted Otherwise)

---

### AREA OF STEEL REQUIRED

<table>
<thead>
<tr>
<th>Section</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>0.00</td>
<td>3.00</td>
<td>6.00</td>
<td>9.00</td>
<td>12.00</td>
</tr>
<tr>
<td>As(Longitudinal)</td>
<td>0.02</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.02</td>
</tr>
<tr>
<td>As/sv(Trans Y)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>As/sv(Trans Z)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

### AREA OF STEEL REQUIRED

<table>
<thead>
<tr>
<th>Section</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>0.00</td>
<td>3.00</td>
<td>6.00</td>
<td>9.00</td>
<td>12.00</td>
</tr>
<tr>
<td>As(Longitudinal)</td>
<td>0.06</td>
<td>0.02</td>
<td>0.01</td>
<td>0.01</td>
<td>0.05</td>
</tr>
<tr>
<td>As/sv(Trans Y)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>As/sv(Trans Z)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

60. END CONCRETE DESIGN

61. FINISH

******** END OF THE STAAD.Pro RUN ********

**** DATE= APR 14, 2019 TIME= 22:57:35 ****
EX. US-9 Modeling Slabs and Shear Walls Using Finite Elements

The space frame structure in this example consists of frame members and finite elements (plates). The finite element part is used to model floor slabs and a shear wall. Concrete design of an element is performed.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-9 Modeling Slabs and Shear Walls Using Finite Elements.STD when you install the program.

Figure 462: Example Problem No. 9

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE
  * EXAMPLE PROBLEM WITH FRAME MEMBERS AND FINITE ELEMENTS

UNIT FEET KIP

The units for the data that follows are specified above.

JOINT COORD
  1 0 0 0 ; 2 0 0 20
```
The joint numbers and their coordinates are defined through the above set of commands. The automatic
generation facility has been used several times in the above lines. See section 5.11 of the Technical Reference
Manual on page 2425 where the joint coordinate generation facilities are described.

The member incidences are defined through the above set of commands. For some members, the member
number followed by the start and end joint numbers are defined. In other cases, STAAD's automatic generation
facilities are utilized. Section 5 of the Technical Reference Manual describes these facilities in detail.

The above lines define the nodes of super-elements. Super-elements are plate/shell surfaces from which a
number of individual plate/shell elements can be generated. In this case, the points describe the outer corners of
a slab and that of a shear wall. Our goal is to define the slab and the wall as several plate/shell elements.

The above lines form the instructions to generate individual 4-noded elements from the super-element profiles.
For example, the command MESH ABCD 4 4 means that STAAD.Pro has to generate 16 elements from the
surface formed by the points A, B, C and D with 4 elements along the edges AB & CD and 4 elements along the edges BC & DA.

<table>
<thead>
<tr>
<th>MEMB PROP</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 40 PRIS YD 1 ZD 1</td>
</tr>
</tbody>
</table>

Members 1 to 40 are defined as a rectangular prismatic section with 1 ft depth and 1 ft width.

<table>
<thead>
<tr>
<th>ELEM PROP</th>
</tr>
</thead>
<tbody>
<tr>
<td>41 TO 88 TH 0.5</td>
</tr>
</tbody>
</table>

Elements 41 to 88 are defined to be 0.5 ft thick.

<table>
<thead>
<tr>
<th>UNIT INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOIsoetropic CONCRETE</td>
</tr>
<tr>
<td>E 3000</td>
</tr>
<tr>
<td>POISSON 0.17</td>
</tr>
<tr>
<td>DENSITY 8.7e-005</td>
</tr>
<tr>
<td>ALPHA 5e-006</td>
</tr>
<tr>
<td>DAMP 0.05</td>
</tr>
<tr>
<td>G 1346.15</td>
</tr>
<tr>
<td>TYPE CONCRETE</td>
</tr>
<tr>
<td>STRENGTH FCU 4</td>
</tr>
<tr>
<td>END DEFINE MATERIAL</td>
</tr>
<tr>
<td>CONSTANTS</td>
</tr>
<tr>
<td>MATERIAL CONCRETE ALL</td>
</tr>
</tbody>
</table>

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

<table>
<thead>
<tr>
<th>SUPPORT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 6 FIXED</td>
</tr>
</tbody>
</table>

Joints 1 to 6 are defined as fixed supported.

<table>
<thead>
<tr>
<th>UNIT FEET</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD 1 DEAD LOAD FROM FLOOR</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
</tr>
<tr>
<td>41 TO 72 PRESSURE -1.0</td>
</tr>
</tbody>
</table>

Load 1 consists of a pressure load of 1 Kip/sq.ft. intensity on elements 41 to 72. The negative sign (and the default value for the axis) indicates that the load acts opposite to the positive direction of the element local z-axis.

<table>
<thead>
<tr>
<th>LOAD 2 WIND LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD</td>
</tr>
<tr>
<td>11 33 FZ -20.</td>
</tr>
<tr>
<td>22 FZ -100.</td>
</tr>
</tbody>
</table>

Load 2 consists of joint loads in the Z direction at joints 11, 22, and 33.

<table>
<thead>
<tr>
<th>LOAD COMB 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0.9 2 1.3</td>
</tr>
</tbody>
</table>

Load 3 is a combination of 0.9 times load case 1 and 1.3 times load case 2.

<table>
<thead>
<tr>
<th>PERFORM ANALYSIS</th>
</tr>
</thead>
</table>

The command to perform an elastic analysis is specified above.

<table>
<thead>
<tr>
<th>LOAD LIST 1 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRINT SUPP REAC</td>
</tr>
</tbody>
</table>
Support reactions, members forces and element stresses are printed for load cases 1 and 3.

The above set of command form the instructions to STAAD to perform a concrete design on element 47. Design is done according to the ACI 318 code. Note that design will consist only of flexural reinforcement calculations in the longitudinal and transverse directions of the elements for the moments MX and MY.

The STAAD run is terminated.
E JOINT 33
F JOINT 29
G JOINT 3
H JOINT 4
GENERATE ELEMENT
MESH ABCD 4 4
MESH DCEF 4 4
MESH DCHG 4 4
MEMB PROP
1 TO 40 PRIS YD 1 ZD 1
ELEM PROP
41 TO 88 TH 0.5
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3000
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 6 FIXED
UNIT FEET
LOAD 1 DEAD LOAD FROM FLOOR
ELEMENT LOAD
41 TO 72 PRESSURE -1.0
LOAD 2 WIND LOAD
JOINT LOAD
11 33 FZ -20.
22 FZ -100.
LOAD COMB 3
1 0.9 2 1.3
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT SUPP REAC
PRINT MEMBER FORCES LIST 27
PRINT ELEMENT STRESSES LIST 47
START CONCRETE DESIGN
CODE ACI
DESIGN ELEMENT 47
END CONCRETE DESIGN
FINI
1. STAAD SPACE

INPUT FILE: US-9 Modeling Slabs and Shear Walls Using Finite Elements.STD

2. * EXAMPLE PROBLEM WITH FRAME MEMBERS AND

3. * FINITE ELEMENTS

4. UNIT FEET KIP

5. JOINTCOORD

6. 1 0 0 0 ; 2 0 0 20

7. REP ALL 2 20 0 0

8. 7 0 15 0 11 0 15 20

9. 12 5 15 0 14 15 15 0

10. 15 5 15 20 17 15 15 20

11. 18 20 15 0 22 20 15 20

12. 23 25 15 0 25 35 15 0

13. 26 25 15 20 28 35 15 20

14. 29 40 15 0 33 20 15 20

15. 34 20 3.75 0 36 20 11.25 0

16. 37 20 3.75 20 39 20 11.25 20

17. MEMBER INCI

18. *COLUMNS

19. 1 1 7 ; 2 2 11

20. 3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18

21. 7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22

22. 11 5 29 ; 12 6 33

23. *BEAMS IN Z DIRECTION AT X=0

24. 13 7 8 16

25. *BEAMS IN Z DIRECTION AT X=20

26. 17 18 19 20

27. *BEAMS IN Z DIRECTION AT X=40

28. 21 29 30 24

29. *BEAMS IN X DIRECTION AT Z = 0

30. 25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18

31. 29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29

32. *BEAMS IN X DIRECTION AT Z = 20

33. 33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22

34. 37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33

35. DEFINE MESH

36. A JOINT 7

37. B JOINT 11

38. C JOINT 22

-- PAGE NO. 2

39. D JOINT 18

40. E JOINT 33

41. F JOINT 29

42. G JOINT 3

43. H JOINT 4

44. GENERATE ELEMENT

45. MESH ABCD 4 4

46. MESH DCEF 4 4

47. MESH DCHG 4 4

48. MEMB PROP

49. 1 TO 40 PRIS YD 1 ZD 1

50. ELEM PROP
## Application Examples

**EX. American Design Examples**

51. 41 TO 88 TH 0.5  
52. UNIT INCH  
53. DEFINE MATERIAL START  
54. ISOTROPIC CONCRETE  
55. E 3000  
56. POISSON 0.17  
57. DENSITY 8.7E-005  
58. ALPHA 5E-006  
59. DAMP 0.05  
60. G 1346.15  
61. TYPE CONCRETE  
62. STRENGTH FCU 4  
63. END DEFINE MATERIAL  
64. CONSTANTS  
65. MATERIAL CONCRETE ALL  
66. SUPPORT  
67. 1 TO 6 FIXED  
68. UNIT FEET  
69. LOAD 1 DEAD LOAD FROM FLOOR  
70. ELEMENT LOAD  
71. 41 TO 72 PRESSURE -1.0  
72. LOAD 2 WIND LOAD  
73. JOINT LOAD  
74. 11 33 FZ -20.  
75. 22 FZ -100.  
76. LOAD COMB 3  
77. 1 0.9 2 1.3  
78. PERFORM ANALYSIS

### Problem Statistics

<table>
<thead>
<tr>
<th>Number of joints</th>
<th>Number of members</th>
<th>Number of plates</th>
<th>Number of solids</th>
<th>Number of supports</th>
</tr>
</thead>
<tbody>
<tr>
<td>69</td>
<td>40</td>
<td>48</td>
<td>0</td>
<td>6</td>
</tr>
</tbody>
</table>

Using 64-bit analysis engine.

---

**Problem Statistics**

- Number of joints: 69
- Number of members: 40
- Number of plates: 48
- Number of solids: 0
- Number of supports: 6

TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 378
TOTAL LOAD COMBINATION CASES = 1 SO FAR.

79. LOAD LIST 1 3  
80. PRINT SUPP REAC  
SUPP REAC  

---

**Support Reactions - Unit Kip Feet**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>9.11</td>
<td>82.56</td>
<td>11.46</td>
<td>56.95</td>
<td>-0.01</td>
<td>-45.28</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>8.23</td>
<td>74.62</td>
<td>10.65</td>
<td>54.04</td>
<td>0.03</td>
<td>-40.81</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>9.11</td>
<td>82.56</td>
<td>-11.46</td>
<td>-56.95</td>
<td>0.01</td>
<td>-45.28</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>8.19</td>
<td>73.98</td>
<td>-9.97</td>
<td>-48.33</td>
<td>0.19</td>
<td>-40.77</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>234.89</td>
<td>75.78</td>
<td>-39.25</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>234.89</td>
<td>-75.78</td>
<td>39.25</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>77.69</td>
<td>20.17</td>
<td>50.98</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-9.11</td>
<td>82.56</td>
<td>11.46</td>
<td>56.95</td>
<td>0.01</td>
<td>45.28</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-8.23</td>
<td>74.62</td>
<td>10.65</td>
<td>54.04</td>
<td>0.03</td>
<td>40.81</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-9.11</td>
<td>82.56</td>
<td>-11.46</td>
<td>-56.95</td>
<td>-0.01</td>
<td>45.28</td>
</tr>
</tbody>
</table>
3   -8.19   73.98   -9.97   -48.33   -0.19   40.77
*************** END OF LATEST ANALYSIS RESULT ***************
81. PRINT MEMBER FORCES LIST 27
MEMBER FORCES LIST 27
STAAD SPACE -- PAGE NO. 5
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
MEMBER END FORCES STRUCTURE TYPE = SPACE
-----------------
ALL UNITS ARE -- KIP FEET (LOCAL )
MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
27 1 13 0.73 -13.21 -0.07 25.88 0.18 -80.00
14 -0.73 13.21 0.07 -25.88 0.18 13.96
3 13 5.42 -11.85 -0.21 23.44 0.51 -72.04
14 -5.42 11.85 0.21 -23.44 0.53 12.79
*************** END OF LATEST ANALYSIS RESULT ***************
82. PRINT ELEMENT STRESSES LIST 47
ELEMENT STRESSES LIST 47
STAAD SPACE -- PAGE NO. 6
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
ELEMENT STRESSES FORCE,LENGTH UNITS= KIP FEET
-----------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
ELEMENT LOAD SQX SQY MX MY MXY
TRESCAT TRESPAB
47 1 1.71 0.50 -11.52 -14.78 1.47
330.22 326.90 -1.21 -1.70 0.52
370.35 366.37
TOP : SMAX= -263.74 SMIN= -370.35 TMAX= 53.30 ANGLE= 21.2
BOTT: SMAX= 366.37 SMIN= 261.89 TMAX= 52.24 ANGLE= -69.1
3 1.53 0.45 -10.40 -13.30 1.29
299.11 292.24 -1.83 -2.46 4.00
336.09 326.02
TOP : SMAX= -236.95 SMIN= -336.09 TMAX= 49.57 ANGLE= 22.4
BOTT: SMAX= 326.02 SMIN= 238.44 TMAX= 43.79 ANGLE= -71.0
**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
MAXIMUM PRINCIPAL PRINCIPAL SHEAR VONMISES TRESPA
MINIMUM PRINCIPAL PRINCIPAL SHEAR VONMISES TRESPA
STRESS STRESS STRESS STRESS STRESS
3.663715E+02 3.703516E+02 5.330365E+01 3.302169E+02 3.703516E+02
PLATE NO. 47 47 47 47 47
CASE NO. 1 1 1 1 1
*************** END OF ELEMENT FORCES ***********************
83. START CONCRETE DESIGN
STAAD SPACE -- PAGE NO. 7
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
CONCRETE DESIGN
84. CODE ACI
85. DESIGN ELEMENT 47
STAAD SPACE -- PAGE NO. 8
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
**WARNING** - ELEMENT DESIGN to ACI 318-14 is not available in this version
of STAAD.Pro
86. END CONCRETE DESIGN
87. FINI
************ END OF THE STAAD.Pro RUN ************
**** DATE= APR 14,2019 TIME= 22:57:39 ****
********************************************************************
EX. US-10 Finite Element Model for a Rectangular Tank

A tank structure is modeled with four-noded plate elements. Water pressure from inside is used as loading for the tank. Reinforcement calculations have been done for some elements.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-10 Finite Element Model for a Rectangular Tank.STD when you install the program.

Figure 463: Tank Model
Application Examples

EX. American Design Examples

Figure 464: Deflected Shape

Actual input is shown in bold lettering followed by explanation.

<table>
<thead>
<tr>
<th>STAAD SPACE FINITE ELEMENT MODEL OF TANK</th>
</tr>
</thead>
<tbody>
<tr>
<td>* STRUCTURE</td>
</tr>
</tbody>
</table>

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D) structure.

<table>
<thead>
<tr>
<th>UNITS FEET KIPS</th>
</tr>
</thead>
</table>

Defines the input units for the data that follows.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0. 0. 0. 5 0. 20. 0.</td>
</tr>
<tr>
<td>REPEAT 4 5. 0. 0.</td>
</tr>
<tr>
<td>REPEAT 4 0. 0. 5.</td>
</tr>
<tr>
<td>REPEAT 4 -5. 0. 0.</td>
</tr>
<tr>
<td>REPEAT 3 0. 0. -5.</td>
</tr>
<tr>
<td>81 5. 0. 5. 83 5. 0. 15.</td>
</tr>
<tr>
<td>REPEAT 2 5. 0. 0.</td>
</tr>
</tbody>
</table>

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command generates joint coordinates by repeating the pattern of the previous line of joint coordinates. The number following the REPEAT command is the number of repetitions to be carried out. This is followed by X, Y and Z coordinate increments. See section 5.11 of the Technical Reference Manual (on page 2425).

<table>
<thead>
<tr>
<th>ELEMENT INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 7 6 TO 4 1 1</td>
</tr>
<tr>
<td>REPEAT 14 4 5</td>
</tr>
<tr>
<td>61 76 77 2 1 TO 64 1 1</td>
</tr>
<tr>
<td>65 1 6 81 76</td>
</tr>
<tr>
<td>66 76 81 82 71</td>
</tr>
<tr>
<td>67 71 82 83 66</td>
</tr>
<tr>
<td>68 66 83 56 61</td>
</tr>
<tr>
<td>69 6 11 84 81</td>
</tr>
<tr>
<td>70 81 84 85 82</td>
</tr>
<tr>
<td>71 82 85 86 83</td>
</tr>
<tr>
<td>72 83 86 51 56</td>
</tr>
<tr>
<td>73 11 16 87 84</td>
</tr>
<tr>
<td>74 84 87 88 85</td>
</tr>
<tr>
<td>75 85 88 89 86</td>
</tr>
<tr>
<td>76 86 89 46 51</td>
</tr>
<tr>
<td>77 16 21 26 87</td>
</tr>
</tbody>
</table>
Element connectivities are input as above by providing the element number followed by joint numbers defining the element. The REPEAT command generates element incidences by repeating the pattern of the previous line of element nodes. The number following the REPEAT command is the number of repetitions to be carried out and that is followed by element and joint number increments. See section 5.12 of the Technical Reference Manual (on page 2428).

UNIT INCHES
ELEMENT PROPERTIES
1 TO 80 TH 8.0

Element properties are provided by specifying that the elements are 8.0 inches THick.

DEFINE MATERIAL ISOTROPIC CONCRETE
E 3000.
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

SUPPORT
1 TO 76 BY 5 81 TO 89 PINNED

Pinned supports are specified at the joints listed above. No moments will be carried by these supports. The expression 1 TO 76 BY 5 means 1, 6, 11, etc. up to 76.

UNIT FT
LOAD 1
ELEMENT LOAD
4 TO 64 BY 4 PR 1.
3 TO 63 BY 4 PR 2.
2 TO 62 BY 4 PR 3.
1 TO 61 BY 4 PR 4.

Load case 1 is initiated. It consists of element loads in the form of uniform pressure(indicated by PR) acting along the local z-axis.

PERFORM ANALYSIS

This command instructs the program to proceed with the analysis.

UNIT INCHES
PRINT JOINT DISPLACEMENTS LIST 5 25 45 65

The joint displacement values for the listed nodes will be reported in the output file as a result of the above command.

PRINT ELEM FORCE LIST 13 16
PRINT ELEM STRESS LIST 9 12
Two types of results are requested for elements. The first one requests the nodal point forces in the global axes directions to be reported for elements 13 and 16. The second one requests element centroid stresses in the element local axes directions to be reported for elements 9 and 12. These results will appear in a tabular form in the output file.

```
START CONCRETE DESIGN
```

The above command initiates concrete design.

```
CODE ACI
DESIGN SLAB  9  12
```

Slabs (i.e., elements) 9 and 12 will be designed and the reinforcement requirements obtained. In STAAD, elements are typically designed for the moments MX and MY at the centroid of the element.

```
END CONCRETE DESIGN
```

Terminates the concrete design operation.

```
FINISH
```

This command terminates the STAAD run.

### Input File

```
STAAD SPACE FINITE ELEMENT MODEL OF TANK STRUCTURE
UNITS FEET KIPS
JOINT COORDINATES
1 0. 0. 0. 5 0. 20. 0.
REPEAT 4 5. 0. 0.
REPEAT 4 0. 0. 5.
REPEAT 4 -5. 0. 0.
REPEAT 3 0. 0. -5.
81 5. 0. 5. 83 5. 0. 15.
REPEAT 2 5. 0. 0.
ELEMENT INCIDENCES
1 1 2 7 6 TO 4 1 1
REPEAT 14 4 5
61 76 77 2 1 TO 64 1 1
65 1 6 81 76
66 76 81 82 71
67 71 82 83 66
68 66 83 56 61
69 6 11 84 81
70 81 84 85 82
71 82 85 86 83
72 83 86 51 56
73 11 16 87 84
74 84 87 88 85
75 85 88 89 86
76 86 89 46 51
77 16 21 26 87
78 87 26 31 88
79 88 31 36 89
80 89 36 41 46
UNIT INCHES
ELEMENT PROPERTIES
1 TO 80 TH 8.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
```
E 3000.
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 76 BY 5 81 TO 89 PINNED
UNIT FT
LOAD 1
ELEMENT LOAD
4 TO 64 BY 4 PR 1.
3 TO 63 BY 4 PR 2.
2 TO 62 BY 4 PR 3.
1 TO 61 BY 4 PR 4.
PERFORM ANALYSIS
UNIT INCHES
PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
PRINT ELEM FORCE LIST 13 16
PRINT ELEM STRESS LIST 9 12
START CONCRETE DESIGN
CODE ACI
DESIGN SLAB 9 12
END CONCRETE DESIGN
FINISH
13. REPEAT 14 4 5
14. 61 76 77 2 1 TO 64 1 1
15. 65 1 6 81 76
16. 66 76 81 82 71
17. 67 71 82 83 66
18. 68 66 83 56 61
19. 69 6 11 84 81
20. 70 81 84 85 82
21. 71 82 85 86 83
22. 72 83 86 51 56
23. 73 11 16 87 84
24. 74 84 87 88 85
25. 75 85 88 89 86
26. 76 86 89 46 51
27. 77 16 21 26 87
28. 78 87 26 31 88
29. 79 88 31 36 89
30. 80 89 36 41 46
31. UNIT INCHES
32. ELEMENT PROPERTIES
33. 1 TO 80 TH 8.0
34. DEFINE MATERIAL START
35. ISOTROPIC CONCRETE
36. E 3000.
37. POISSON 0.17
38. DENSITY 8.7E-005
39. ALPHA 5E-006
40. DAMP 0.05
41. G 1346.15
42. TYPE CONCRETE
43. STRENGTH FCU 4
44. END DEFINE MATERIAL
45. CONSTANTS
46. MATERIAL CONCRETE ALL
47. SUPPORT
48. 1 TO 76 BY 5 81 TO 89 PINNED
49. UNIT FT
50. LOAD 1
51. ELEMENT LOAD
52. 4 TO 64 BY 4 PR 1.
53. 3 TO 63 BY 4 PR 2.
54. 2 TO 62 BY 4 PR 3.
55. 1 TO 61 BY 4 PR 4.
56. PERFORM ANALYSIS

PROBLEM STATISTICS

NUMBER OF JOINTS 89
NUMBER OF MEMBERS 0
NUMBER OF PLATES 80
NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0
NUMBER OF SUPPORTS 25

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 459
TOTAL LOAD COMBINATION CASES = 0 SO FAR.
57. UNIT INCHES
58. PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
JOINT DISPLACE LIST 5
FINITE ELEMENT MODEL OF TANK STRUCTURE

-- PAGE NO. 2
JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = SPACE

------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
5    1   -0.00403   0.00027  -0.00403   0.00021   0.00000  -0.00021
25    1    0.00403   0.00027  -0.00403   0.00021   0.00000   0.00021
45    1    0.00403   0.00027   0.00403  -0.00021   0.00000   0.00021
65    1   -0.00403   0.00027   0.00403  -0.00021   0.00000  -0.00021

*************** END OF LATEST ANALYSIS RESULT ***************

59. PRINT ELEM FORCE LIST 13 16

ELEM FORCE LIST

FINITE ELEMENT MODEL OF TANK STRUCTURE -- PAGE NO. 4

ELEMENT FORCES  FORCE,LENGTH UNITS= KIPS INCH

---------------
GLOBAL CORNER FORCES

JOINT     FX          FY          FZ          MX          MY          MZ
ELE.NO.     13 FOR LOAD CASE     1
16  2.3570E+01  -1.8550E+01  -3.9109E+00  -1.3649E+02   2.4437E+02   1.3427E+02
17  7.6147E+00  -1.5328E+01  -1.6950E+01  -8.5004E+01   -4.0990E+01  -2.1558E+02
22  4.0892E+01   5.9984E+00   1.8154E+01  -1.4891E+02   5.4757E+02   2.6486E+02
21  1.0308E+01   2.7879E+01   2.7070E+00  -2.9817E+02   5.0071E+02  -1.8356E+02

---------------
GLOBAL CORNER FORCES

JOINT     FX          FY          FZ          MX          MY          MZ
ELE.NO.     16 FOR LOAD CASE     1
19  -3.7438E+01   3.4537E+00  -2.8252E+01  -2.4192E+02   1.9474E+02   2.4055E+02
20  -3.6565E+01   6.0104E-01  -2.7049E+01  2.5567E+02  -2.1788E+02   4.0380E+02
25  3.2510E+01   1.7764E-15   2.6260E+01  -4.9347E+02   1.4655E+03   4.9347E+02
24  4.1493E+01  -4.0547E+00   2.9041E+01  5.2703E+02  1.4655E+03  -3.3022E+02

60. PRINT ELEM STRESS LIST 9 12

ELEM STRESS LIST

FINITE ELEMENT MODEL OF TANK STRUCTURE -- PAGE NO. 5

ELEMENT STRESSES  FORCE,LENGTH UNITS= KIPS INCH

----------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
VONT       VONB         SX          SY          SXY
TRESCAT    TRESCAB
9      1         0.15        0.01       -5.43        5.56        5.60
1.35        1.20        0.02        0.08        0.03
1.55        1.39
TOP: SMAX= 0.83 SMIN= -0.72 TMAX= 0.78 ANGLE= 67.3
BOTT: SMAX= 0.74 SMIN= -0.65 TMAX= 0.69 ANGLE= -22.8
12      1        -0.02       -0.04       -0.11       20.01       -0.21
2.05        1.72       -0.00       0.16        0.01
2.05        1.72
TOP: SMAX= 2.04 SMIN= -0.01 TMAX= 1.03 ANGLE= -89.3
BOTT: SMAX= 0.01 SMIN= -1.71 TMAX= 0.86 ANGLE= 0.4

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
MAXIMUM PRINCIPAL  MINIMUM PRINCIPAL  MAXIMUM SHEAR VONMISES TRESCAT
STRESS  STRESS  STRESS  STRESS  STRESS
2.040916E+00  -1.712074E+00  1.026498E+00  2.046982E+00  2.052995E+00

PLATE NO.  CASE NO.  12 12 12 12 12
1 1 1 1 1

***********************END OF ELEMENT FORCES***********************

61. START CONCRETE DESIGN

FINITE ELEMENT MODEL OF TANK STRUCTURE -- PAGE NO. 6

CONCRETE DESIGN

62. CODE ACI

63. DESIGN SLAB 9 12

FINITE ELEMENT MODEL OF TANK STRUCTURE -- PAGE NO. 7
Related Links

- TR.14.2 Element Mesh Generation (on page 2435)

EX. US-11 Response Spectrum Analysis of a Frame

Dynamic analysis (Response Spectrum) is performed for a steel structure. Results of a static and dynamic analysis are combined. The combined results are then used for steel design.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-11 Response Spectrum Analysis of a Frame.STD when you install the program.
Where:

$L_1 = 10$ ft, $L_2 = 20$ ft

$w = 1.5$ k/ft

$P_1 = 5.0$ kips, $P_2 = 7.5$ kips

Actual input is shown in bold lettering followed by explanation.

**STAAD PLANE RESPONSE SPECTRUM ANALYSIS**

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

**UNIT FEET KIPS**

Defines the input units for the data that follows.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0 0 0 ; 2 20 0 0</td>
</tr>
<tr>
<td>3 0 10 0 ; 4 20 10 0</td>
</tr>
<tr>
<td>5 0 20 0 ; 6 20 20 0</td>
</tr>
</tbody>
</table>

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are all the same, in this case, zeros.
**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6</td>
</tr>
<tr>
<td>5 3 4 ; 6 5 6</td>
</tr>
</tbody>
</table>

Defines the members by the joints to which they are connected.

<table>
<thead>
<tr>
<th>MEMBER PROPERTIES AMERICAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 4 TA ST W10X33</td>
</tr>
<tr>
<td>5 TA ST W12X40</td>
</tr>
<tr>
<td>6 TA ST W8X40</td>
</tr>
</tbody>
</table>

Properties for all members are assigned from the American (AISC) steel table. The word ST stands for standard single section.

<table>
<thead>
<tr>
<th>SUPPORTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 2 FIXED</td>
</tr>
</tbody>
</table>

Fixed supports are specified at joints 1 and 2.

<table>
<thead>
<tr>
<th>UNIT INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td>E 29000.</td>
</tr>
<tr>
<td>POISSON 0.3</td>
</tr>
<tr>
<td>DENSITY 283e-006</td>
</tr>
<tr>
<td>ALPHA 6e-006</td>
</tr>
<tr>
<td>DAMP 0.03</td>
</tr>
<tr>
<td>TYPE STEEL</td>
</tr>
<tr>
<td>STRENGTH FY 36 FU 58 RY 1.5 RT 1.2</td>
</tr>
<tr>
<td>END DEFINE MATERIAL</td>
</tr>
<tr>
<td>CONSTANT</td>
</tr>
<tr>
<td>MATERIAL STEEL ALL</td>
</tr>
</tbody>
</table>

Material constants such as E (modulus of elasticity), Poisson’s ratio and density (DEN) are specified above. Length unit is changed from FT to INCH to facilitate the input.

<table>
<thead>
<tr>
<th>CUT OFF MODE SHAPE 2</th>
</tr>
</thead>
</table>

The number of mode shapes to be considered in dynamic analysis is set to 2. Without the above command, this will be set to the default. See section 5.30.1 of the Technical Reference Manual (on page 2539).

<table>
<thead>
<tr>
<th>* LOAD 1 WILL BE STATIC LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNIT FEET</td>
</tr>
<tr>
<td>LOAD 1 DEAD AND LIVE LOADS</td>
</tr>
</tbody>
</table>

Load case 1 is initiated followed by a title. Prior to this, the length unit is changed to METER for specifying distributed member loads. A line starting with an asterisk (*) mark indicates a comment line.

<table>
<thead>
<tr>
<th>SELFWEIGHT Y -1.0</th>
</tr>
</thead>
</table>

The above command indicates that the selfweight of the structure acting in the global Y direction is part of this load case. The factor of -1.0 is meant to indicate that the load acts opposite to the positive direction of global Y, hence downwards.
Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The word UNI stands for uniformly distributed load while CON stands for concentrated load. GY is followed by the value of the load and the distance at which it is applied.

* NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
  * WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 2 SEISMIC LOADING

The two lines which begin with the asterisk are comment lines which tell us the purpose of the next load case. Load case 2 is then initiated along with an optional title. This will be a dynamic load case. Permanent masses will be provided in the form of loads. These masses (in terms of loads) will be considered for the eigensolution. Internally, the program converts these loads to masses, hence it is best to specify them as absolute values (without a negative sign). Also, the direction (X, Y, Z etc.) of the loads will correspond to the dynamic degrees of freedom in which the masses are capable of vibrating. In a PLANE frame, only X and Y directions need to be considered. In a SPACE frame, masses (loads) should be provided in all three (X, Y and Z) directions if they are active along all three. The user has the freedom to restrict one or more directions.

SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0

The above commands indicate that the selfweight of the structure acting in the global X and Y directions with a factor of 1.0 is taken into consideration for the mass matrix.

MEMBER LOADS

<table>
<thead>
<tr>
<th>Load</th>
<th>Load Type</th>
<th>Gx Value</th>
<th>Gy Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>CON</td>
<td>5.0</td>
<td>6.0</td>
</tr>
<tr>
<td>5</td>
<td>CON</td>
<td>5.0</td>
<td>6.0</td>
</tr>
<tr>
<td>5</td>
<td>CON</td>
<td>7.5</td>
<td>10.0</td>
</tr>
<tr>
<td>5</td>
<td>CON</td>
<td>7.5</td>
<td>10.0</td>
</tr>
<tr>
<td>5</td>
<td>CON</td>
<td>5.0</td>
<td>14.0</td>
</tr>
<tr>
<td>5</td>
<td>CON</td>
<td>5.0</td>
<td>14.0</td>
</tr>
</tbody>
</table>

The mass matrix will also consist of terms derived from the above member loads. GX and GY indicate that the load, and hence the resulting mass, is capable of vibration along the global X and Y directions. The word CON stands for concentrated load. Concentrated forces of 5, 7.5, and 5 kips are located at 6ft, 10ft and 14ft from the start of member 5.

SPECTRUM CQC IBC 2012 X 1.0 ACC DAMP 0.05
ZIP 92806 SITE CLASS D TL 12.0

The SPECTRUM command specifies a 2012 International Building Code / ASCE 7-10 seismic response spectrum load. The modal responses will be combined using the CQC method. Here, the spectrum effect is in the global X direction with a factor of 1.0. IBC 2012 response spectra are always given in terms of acceleration (ACC). A damping ratio of 0.05 (5%) is used. The second line then gives the location and site class values (using the US ZIP postal code and IBC/ASCE site class) along with the long period transition time.

LOAD LIST  1  3  4
PARAMETER CODE AISC UNIFIED SELECT ALL

A steel design in the form of a member selection is performed based on the rules of the American Code. Only the member forces resulting from load cases 1, 3 and 4 will be considered for these calculations.

FINISH

This command terminates the STAAD run.
Application Examples
EX. American Design Examples

Input File

STAAAD PLANE RESPONSE SPECTRUM ANALYSIS
UNIT FEET KIPS
JOINT COORDINATES
1 0 0 0 ; 2 20 0 0
3 0 10 0 ; 4 20 10 0
5 0 20 0 ; 6 20 20 0
MEMBER INCIDENCES
1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
5 3 4 ; 6 5 6
MEMBER PROPERTIES AMERICAN
1 TO 4 TA ST W10X33
5 TA ST W12X40
6 TA ST W8X40
SUPPORTS
1 2 FIXED
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
CUT OFF MODE SHAPE 2
*LOAD 1 WILL BE STATIC LOAD
UNIT FEET
LOAD 1 DEAD AND LIVE LOADS
SELFWEIGHT Y -1.0
MEMBER LOADS
5 CON GY -5.0 6.0
5 CON GY -7.5 10.0
5 CON GY -5.0 14.0
5 6 UNI Y -1.5
* NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
* WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 2 SEISMIC LOADING
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GX 5.0 6.0
5 CON GX 7.5 10.0
5 CON GX 7.5 10.0
5 CON GX 5.0 14.0
5 CON GX 5.0 14.0
SPECTRUM CQC IBC 2012 X 1.0 ACC DAMP 0.05
ZIP 92806 SITE CLASS D TL 12.0
LOAD COMBINATION 3
1 0.75 2 0.75
LOAD COMBINATION 4
1 0.75 2 -0.75
PERFORM ANALYSIS PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
LOAD LIST 1 3 4
PARAMETER
CODE AISC UNIFIED
SELECT ALL
FINISH

STAAD Output File

1. STAAD PLANE RESPONSE SPECTRUM ANALYSIS

INPUT FILE: US-11 Response Spectrum Analysis of a Frame.STD

2. UNIT FEET KIPS
3. JOINT COORDINATES
4. 1 0 0 0 ; 2 20 0 0
5. 3 0 10 0 ; 4 20 10 0
6. 5 0 20 0 ; 6 20 20 0
7. MEMBER INCIDENCES
8. 1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
9. 5 4 6 ; 6 5 6
10. MEMBER PROPERTIES AMERICAN
11. 1 TO 4 TA ST W10X33
12. 5 TA ST W12X40
13. 6 TA ST W8X40
14. SUPPORTS
15. 1 2 FIXED
16. UNIT INCH
17. DEFINE MATERIAL START
18. ISOTROPIC STEEL
19. E 29000.
20. POISSON 0.3
21. DENSITY 283E-006
22. ALPHA 6E-006
23. DAMP 0.03
24. TYPE STEEL
25. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
26. END DEFINE MATERIAL
27. CONSTANT
28. MATERIAL STEEL ALL
29. CUT OFF MODE SHAPE 2
30. *LOAD 1 WILL BE STATIC LOAD
31. UNIT FEET
32. LOAD 1 DEAD AND LIVE LOADS
33. SELFWEIGHT Y -1.0
34. MEMBER LOADS
Application Examples
EX. American Design Examples

35. 5 CON GY -5.0  6.0
36. 5 CON GY -7.5 10.0
37. 5 CON GY -5.0 14.0
38. 5 6 UNI Y -1.5
RESPONSE SPECTRUM ANALYSIS
-- PAGE NO. 2
39. * NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
40. * WITH MASSES PROVIDED IN TERMS OF LOAD.
41. LOAD 2 SEISMIC LOADING
42. SELFWEIGHT X 1.0
43. SELFWEIGHT Y 1.0
44. MEMBER LOADS
45. 5 CON GX 5.0 6.0
46. 5 CON GY 5.0 6.0
47. 5 CON GX 7.5 10.0
48. 5 CON GY 7.5 10.0
49. 5 CON GX 5.0 14.0
50. 5 CON GY 5.0 14.0
51. SPECTRUM CQC IBC 2012 X 1.0 ACC DAMP 0.05
52. ZIP 92806 SITE CLASS D TL 12.0
53. LOAD COMBINATION 3
54. 1 0.75 2 0.75
55. LOAD COMBINATION 4
56. 1 0.75 2 -0.75
57. PERFORM ANALYSIS PRINT MODE SHAPES

Problem Statistics
-----------------------------------
NUMBER OF JOINTS 6  NUMBER OF MEMBERS 6
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 12
TOTAL LOAD COMBINATION CASES = 2 SO FAR.
***NOTE: MASSES DEFINED UNDER LOAD# 2 WILL FORM
THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.

Eigen Method: SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED = 2
NUMBER OF EXISTING MASSES IN THE MODEL = 8
NUMBER OF MODES THAT WILL BE USED = 2
*** EIGENSOLUTION: ADVANCED METHOD ***
RESPONSE SPECTRUM ANALYSIS
-- PAGE NO. 3
CALCULATED FREQUENCIES FOR LOAD CASE 2

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.493</td>
<td>0.22258</td>
</tr>
<tr>
<td>2</td>
<td>16.308</td>
<td>0.06132</td>
</tr>
</tbody>
</table>

RESPONSE SPECTRUM ANALYSIS
-- PAGE NO. 4

Mode Shapes
-------------
<table>
<thead>
<tr>
<th>JOINT</th>
<th>MODE</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.67185</td>
<td>0.00308</td>
<td>0.00000</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>-3.367E-03</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.67185</td>
<td>-0.00308</td>
<td>0.00000</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>-3.367E-03</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>1.00000</td>
<td>0.00360</td>
<td>0.00000</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>-1.467E-03</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>1.00000</td>
<td>-0.00360</td>
<td>0.00000</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>-1.467E-03</td>
</tr>
</tbody>
</table>

Mode Shapes
-----------
### Modal Weight (Modal Mass Times g) in Kips

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.9841E+01</td>
<td>2.5455E-33</td>
<td>0.0000E+00</td>
<td>9.6801E+00</td>
</tr>
<tr>
<td>2</td>
<td>2.3739E-01</td>
<td>2.4739E-28</td>
<td>0.0000E+00</td>
<td>1.2728E+00</td>
</tr>
</tbody>
</table>

**CQC**

**Modal Combination Method Used.**

**Dynamic Weight X Y Z**

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0079E+01</td>
<td>2.0079E+01</td>
<td>0.0000E+00</td>
<td>KIPS</td>
<td></td>
</tr>
</tbody>
</table>

**Missing Weight X Y Z**

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>-9.34E-05</td>
<td>-2.0079E+01</td>
<td>0.0000E+00</td>
<td>KIPS</td>
<td></td>
</tr>
</tbody>
</table>

**Modal Weight X Y Z**

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0079E+01</td>
<td>2.4739E-28</td>
<td>0.0000E+00</td>
<td>KIPS</td>
<td></td>
</tr>
</tbody>
</table>

### Modal Base Actions

<table>
<thead>
<tr>
<th>Mode</th>
<th>Period</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
<th>MY</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.223</td>
<td>20.45</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.061</td>
<td>0.19</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Participation Factors

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Summ-X</th>
<th>Summ-Y</th>
<th>Summ-Z</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>98.82</td>
<td>0.00</td>
<td>0.00</td>
<td>98.817</td>
<td>0.000</td>
<td>0.000</td>
<td>20.45</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1.18</td>
<td>0.00</td>
<td>0.00</td>
<td>99.999</td>
<td>0.000</td>
<td>0.000</td>
<td>0.19</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

- **Total SRSS Shear**: 20.46 Kips
- **Total 10PCT Shear**: 20.46 Kips
- **Total ABS Shear**: 20.64 Kips
- **Total CQC Shear**: 20.46 Kips

## 58. Print Analysis Results

### Response Spectrum Analysis

**Analysis Results**

**Joint Displacement (Inch Radians)**

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

### Joint Load X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN

<table>
<thead>
<tr>
<th></th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
### Support Reactions - Unit Kips Feet

**Structure Type: Plane**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>4.57</td>
<td>40.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-14.33</td>
</tr>
<tr>
<td>2</td>
<td>10.23</td>
<td>5.16</td>
<td>0.00</td>
<td>0.00</td>
<td>59.43</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>11.10</td>
<td>34.02</td>
<td>0.00</td>
<td>0.00</td>
<td>33.83</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-4.24</td>
<td>26.28</td>
<td>0.00</td>
<td>0.00</td>
<td>-55.32</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>4.57</td>
<td>40.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>14.33</td>
</tr>
<tr>
<td>2</td>
<td>10.23</td>
<td>5.16</td>
<td>0.00</td>
<td>0.00</td>
<td>59.43</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>4.24</td>
<td>34.02</td>
<td>0.00</td>
<td>0.00</td>
<td>55.32</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-11.10</td>
<td>26.28</td>
<td>0.00</td>
<td>0.00</td>
<td>-33.83</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Member End Forces - Unit Kips Feet

**Structure Type: Plane**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>Joint</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>40.20</td>
<td>-4.57</td>
<td>0.00</td>
<td>0.00</td>
<td>-14.33</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-39.87</td>
<td>4.57</td>
<td>0.00</td>
<td>0.00</td>
<td>31.39</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>5.16</td>
<td>10.23</td>
<td>0.00</td>
<td>0.00</td>
<td>59.43</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5.16</td>
<td>10.23</td>
<td>0.00</td>
<td>0.00</td>
<td>42.85</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>34.02</td>
<td>4.24</td>
<td>0.00</td>
<td>0.00</td>
<td>33.83</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-33.78</td>
<td>-4.24</td>
<td>0.00</td>
<td>0.00</td>
<td>-55.68</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>26.28</td>
<td>-11.10</td>
<td>0.00</td>
<td>-55.32</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-26.03</td>
<td>11.10</td>
<td>0.00</td>
<td>0.00</td>
<td>8.59</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>40.20</td>
<td>4.57</td>
<td>0.00</td>
<td>14.33</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-39.87</td>
<td>-4.57</td>
<td>0.00</td>
<td>0.00</td>
<td>31.39</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>5.16</td>
<td>10.23</td>
<td>0.00</td>
<td>59.43</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>5.16</td>
<td>10.23</td>
<td>0.00</td>
<td>0.00</td>
<td>42.85</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>34.02</td>
<td>11.10</td>
<td>0.00</td>
<td>55.32</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-33.78</td>
<td>-11.10</td>
<td>0.00</td>
<td>0.00</td>
<td>-8.59</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>26.28</td>
<td>-4.24</td>
<td>0.00</td>
<td>-33.83</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-26.03</td>
<td>4.24</td>
<td>0.00</td>
<td>0.00</td>
<td>55.68</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>15.73</td>
<td>8.85</td>
<td>0.00</td>
<td>44.36</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-15.40</td>
<td>8.85</td>
<td>0.00</td>
<td>0.00</td>
<td>-44.14</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.89</td>
<td>0.85</td>
<td>0.00</td>
<td>1.28</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.89</td>
<td>0.85</td>
<td>0.00</td>
<td>0.00</td>
<td>8.86</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>12.46</td>
<td>-6.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-32.31</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-12.21</td>
<td>6.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-39.75</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>11.13</td>
<td>-7.28</td>
<td>0.00</td>
<td>-34.23</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-10.88</td>
<td>7.28</td>
<td>0.00</td>
<td>0.00</td>
<td>-26.46</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>15.73</td>
<td>8.85</td>
<td>0.00</td>
<td>44.36</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Response Spectrum Analysis**

**Member End Forces**

**Structure Type = Plane**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Moment-Y</th>
<th>Moment-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>3</td>
<td>-3.21</td>
<td>14.94</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>25.08</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>3.21</td>
<td>21.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.08</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>8.85</td>
<td>15.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>44.14</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>-8.85</td>
<td>15.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-44.14</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.00</td>
<td>0.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>8.86</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>0.00</td>
<td>0.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>8.86</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>6.64</td>
<td>12.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>39.75</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>-6.64</td>
<td>10.88</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-39.75</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>6.64</td>
<td>10.88</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>26.46</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>-6.64</td>
<td>12.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-26.46</td>
<td></td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

---

The capacity results and intermediate results in the report follow the notations and axes labels as defined in the AISC 360-16 code. The analysis results are reported in STAAD.Pro axis convention and the AISC 360-16 design results are reported in AISC 360-16 code axis convention.

**AISC Spec.**  **STAAD.Pro**  **Description**

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Axis typically parallel to the sections principal major axis.</td>
</tr>
<tr>
<td>Y</td>
<td>Y</td>
<td>Axis typically parallel to the sections principal minor axis.</td>
</tr>
<tr>
<td>Z</td>
<td>X</td>
<td>Longitudinal axis perpendicular to the cross section.</td>
</tr>
</tbody>
</table>

**SECTION FORCES AXIS MAPPING:**

<table>
<thead>
<tr>
<th>AISC Spec.</th>
<th>STAAD.Pro</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pz</td>
<td>FX</td>
<td>Axial force.</td>
</tr>
<tr>
<td>Vy</td>
<td>FY</td>
<td>Shear force along minor axis.</td>
</tr>
<tr>
<td>Vx</td>
<td>FZ</td>
<td>Shear force along major axis.</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST W10X22 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.877</th>
<th>Loadcase: 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location: 10.00</td>
<td>Ref: Eq.H1-3a(H1-1b)</td>
<td>Pz: 33.78 C</td>
<td>Vy: 4.242</td>
<td>Vx: 0.000</td>
</tr>
<tr>
<td>Tz: 0.000</td>
<td>My: 0.000</td>
<td>Mx: 55.68</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST W10X22 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.872</th>
<th>Loadcase: 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location: 0.00</td>
<td>Ref: Eq.H1-3a(H1-1b)</td>
<td>Pz: 34.02 C</td>
<td>Vy: 11.10</td>
<td>Vx: 0.000</td>
</tr>
<tr>
<td>Tz: 0.000</td>
<td>My: 0.000</td>
<td>Mx: 55.32</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST W12X19 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.712</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location: 0.00</td>
<td>Ref: Eq.H1-3a(H1-1b)</td>
<td>Pz: 15.73 C</td>
<td>Vy: -8.850</td>
<td>Vx: 0.000</td>
</tr>
<tr>
<td>Tz: 0.000</td>
<td>My: 0.000</td>
<td>Mx: -44.36</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No.</th>
<th>Profile: ST W12X19 (AISC SECTIONS)</th>
<th>Status: PASS</th>
<th>Ratio: 0.712</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location: 0.00</td>
<td>Ref: Eq.H1-3a(H1-1b)</td>
<td>Pz: 15.73 C</td>
<td>Vy: 8.850</td>
<td>Vx: 0.000</td>
</tr>
<tr>
<td>Tz: 0.000</td>
<td>My: 0.000</td>
<td>Mx: 44.36</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
EX. US-12 Moving Load Generation on a Bridge Deck

This example demonstrates generation of load cases for the type of loading known as a moving load. This type of loading occurs classically when the load-causing units move on the structure, as in the case of trucks on a bridge deck. The mobile loads are discretized into several individual immobile load cases at discrete positions. During this process, enormous number of load cases may be created resulting in plenty of output to be sorted. To avoid looking into a lot of output, the maximum force envelope is requested for a few specific members.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-12 Moving Load Generation on a Bridge Deck.STD when you install the program.
Where:

L1 = 30 ft
L2 = 5 ft

Actual input is shown in bold lettering followed by explanation.

<table>
<thead>
<tr>
<th>STAAD FLOOR A SIMPLE BRIDGE DECK</th>
</tr>
</thead>
</table>

Every input has to start with the term STAAD. The word FLOOR signifies that the structure is a floor structure and the geometry is defined through X and Z axis.

| UNITS FEET KIPS |
Defines the input units for the data that follows.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0 0 0 6 25 0 0</td>
</tr>
<tr>
<td>R 5 0 0 30</td>
</tr>
</tbody>
</table>

Joint number followed by X, Y, and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). The first line generates joints 1 through 6. With the repeat (R) command, the coordinates of the next 30 joints are generated by repeating the pattern of the coordinates of the first 6 joints 5 times with X, Y and Z increments of 0,0, and 9 respectively.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 7 6</td>
</tr>
<tr>
<td>7 1 2 11</td>
</tr>
<tr>
<td>R A 4 11 6</td>
</tr>
<tr>
<td>56 31 32 60</td>
</tr>
</tbody>
</table>

Defines the members by the joints to which they are connected. The fourth number indicates the final member number up to which they will be generated. Repeat all (abbreviated as R A) will create members by repeating the member incidence pattern of the previous 11 members. The number of repetitions to be carried out is provided after the RA command and the member increment and joint increment are defined as 11 and 6 respectively. The fifth line of input defines the member incidences for members 56 to 60.

<table>
<thead>
<tr>
<th>MEMBER PROPERTIES AMERICAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 60 TA ST W12X26</td>
</tr>
</tbody>
</table>

Member properties are assigned from the American AISC table for all members. The word ST stands for standard single section.

<table>
<thead>
<tr>
<th>SUPPORTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 6 31 TO 36 PINNED</td>
</tr>
</tbody>
</table>

Pinned supports are specified at the above joints. A pinned support is one which can resist only translational forces.

<table>
<thead>
<tr>
<th>UNITS INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td>E 29000.</td>
</tr>
<tr>
<td>POISSON 0.3</td>
</tr>
<tr>
<td>DENSITY 283e-006</td>
</tr>
<tr>
<td>ALPHA 6e-006</td>
</tr>
<tr>
<td>DAMP 0.03</td>
</tr>
<tr>
<td>TYPE STEEL</td>
</tr>
<tr>
<td>STRENGTH FY 36 FU 58 RY 1.5 RT 1.2</td>
</tr>
<tr>
<td>END DEFINE MATERIAL</td>
</tr>
<tr>
<td>CONSTANT</td>
</tr>
<tr>
<td>MATERIAL STEEL ALL</td>
</tr>
</tbody>
</table>

The input units are changed from FT to INCH. The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

<table>
<thead>
<tr>
<th>UNIT FEET KIP</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MOVING LOAD</td>
</tr>
<tr>
<td>TYPE 1 LOAD 20. 20. 10. DISTANCE 10. 5. WIDTH 10.0</td>
</tr>
</tbody>
</table>

The characteristics of the vehicle are defined above in METER and KNS units. The above lines represent the first out of two sets of data required in moving load generation. The type number (1) is a label for identification of the load-causing unit, such as a truck. Three axles (20 20 10) are specified with the LOAD command. The spacing between the axles in the direction of movement (longitudinal direction) is specified after the DISTANCE
command. WIDTH is the spacing in the transverse direction, that is, it is the distance between the 2 prongs of an axle of the truck.

LOAD 1

Load case 1 is initiated.

SELF Y -1.0

Selfweight of the structure acting in the negative (due to the factor -1.0) global Y direction is the only component of load case 1.

LOAD GENERATION 10
TYPE 1 7.5 0. 0. ZI 10.

This constitutes the second of the two sets of data required for moving load generation. 10 load cases are generated using the Type 1 vehicle whose characteristics were described earlier. For the first of these load cases, the X, Y and Z location of the reference load (see section 5.31.1 of the Technical Reference Manual (on page 2541)) have been specified after the command TYPE 1. The Z Increment of 10ft denotes that the vehicle moves along the Z direction and the individual positions which are 10ft apart will be used to generate the remaining 9 load cases.

As seen in Section 5.31.1 of the Technical Reference manual, the reference wheel is on the last axle. The first load case which is generated will be the one for which the first axle is just about to enter the bridge. The last load case should be the one for which the last axle is just about to exit the bridge. Thus, the total distance travelled by the reference load will be the length of the vehicle (distance from first axle to last axle) plus the span of the bridge. In this problem, that comes to

$$(10+5) + 150 = 165 \text{ feet}.$$ 

This example uses 10 ft increments and generates 10 load cases.

However, if you want the vehicle to move forward in 15 feet increments (each 15 foot increment will create a discrete position of the truck on the bridge), it would required $(165/15)+1 = 12$ cases to be generated.

PERFORM ANALYSIS PRINT LOAD

The above command instructs the program to proceed with the analysis and print the values and positions of all the generated load cases.

PRINT MAXFORCE ENVELOP LIST 3 41 42

A maximum force envelope consisting of the highest forces for each degree of freedom on the listed members will be written into the output file.

FINISH

This command terminates the STAAD run.

**Input File**

<table>
<thead>
<tr>
<th>STAAD FLOOR A SIMPLE BRIDGE DECK</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNITS FEET KIPS</td>
</tr>
<tr>
<td>JOINT COORDINATES</td>
</tr>
<tr>
<td>1 0 0 0 6 25 0 0</td>
</tr>
<tr>
<td>R 5 0 0 30</td>
</tr>
<tr>
<td>MEMBER INCIDENCES</td>
</tr>
<tr>
<td>1 1 7 6</td>
</tr>
<tr>
<td>7 1 2 11</td>
</tr>
<tr>
<td>R A 4 11 6</td>
</tr>
<tr>
<td>56 31 32 60</td>
</tr>
<tr>
<td>MEMBER PROPERTIES AMERICAN</td>
</tr>
</tbody>
</table>
1 TO 60 TA ST W12X26
SUPPORTS
1 TO 6 31 TO 36 PINNED
UNITS INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
UNIT FEET KIP
DEFINE MOVING LOAD
TYPE 1 LOAD 20 20 10 DISTANCE 10 5 WIDTH 10
LOAD 1
SELF Y -1.0
LOAD GENERATION 10
TYPE 1 7.5 0 0 ZI 10
PERFORM ANALYSIS PRINT LOAD
PRINT MAXFORCE ENVELOP LIST 3 41 42
FINISH

STAAD Output File

****************************************************
*                                                  *
*           STAAD.Pro CONNECT Edition              *
*           Version 22.01.00.**                    *
*           Proprietary Program of                 *
*           Bentley Systems, Inc.                  *
*           Date= APR 14, 2019                    *
*           Time= 22:55:25                        *
*                                                  *
*  Licensed to: Bentley Systems Inc                *
****************************************************

1. STAAD FLOOR A SIMPLE BRIDGE DECK
INPUT FILE: US-12 Moving Load Generation on a Bridge Deck.STD
2. UNITS FEET KIPS
3. JOINT COORDINATES
4. 1 0 0 0 6 25 0 0
5. R 5 0 0 30
6. MEMBER INCIDENCES
7. 1 1 7 6
8. 7 1 2 11
9. R A 4 11 6
10. 56 31 32 60
11. MEMBER PROPERTIES AMERICAN
12. 1 TO 60 TA ST W12X26
13. SUPPORTS
14. 1 TO 6 31 TO 36 PINNED
15. UNITS INCH
16. DEFINE MATERIAL START
## Application Examples

**EX. American Design Examples**

17. ISOTROPIC STEEL
18. E 29000
19. POISSON 0.3
20. DENSITY 283E-006
21. ALPHA 6E-006
22. DAMP 0.03
23. TYPE STEEL
24. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
25. END DEFINE MATERIAL
26. CONSTANT
27. MATERIAL STEEL ALL
28. UNIT FEET KIP
29. DEFINE MOVING LOAD
30. TYPE 1 LOAD 20. 20. 10. DISTANCE 10. 5. WIDTH 10.
31. LOAD 1
32. SELF Y -1.0
33. LOAD GENERATION 10
34. TYPE 1 7.5 0. 0. ZI 10.
35. PERFORM ANALYSIS PRINT LOAD

---

### Problem Statistics

<table>
<thead>
<tr>
<th>Number of Joints</th>
<th>36</th>
<th>Number of Members</th>
<th>60</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Plates</td>
<td>0</td>
<td>Number of Solids</td>
<td>0</td>
</tr>
<tr>
<td>Number of Surfaces</td>
<td>0</td>
<td>Number of Supports</td>
<td>12</td>
</tr>
</tbody>
</table>

Using 64-bit analysis engine.

SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 11, TOTAL DEGREES OF FREEDOM = 96
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

---

A SIMPLE BRIDGE DECK

---

### Loading 1

- SELFWEIGHT Y -1.000

ACTUAL WEIGHT OF THE STRUCTURE = 27.278 KIP

---

### Loading 2

- MEMBER LOAD - UNIT KIP FEET

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>-20.0000 GY</td>
<td>2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-20.0000 GY</td>
<td>2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

### Loading 3

- MEMBER LOAD - UNIT KIP FEET

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-10.0000 GY</td>
<td>20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-10.0000 GY</td>
<td>20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-20.0000 GY 2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-20.0000 GY 2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**LOADING 5**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-10.0000 GY 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**LOADING 6**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-5.0000 GY 25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-5.0000 GY 25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-5.0000 GY 25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-5.0000 GY 25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**LOADING 7**

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-10.0000 GY 20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MEMBER</td>
<td>UDL</td>
<td>L1</td>
<td>L2</td>
<td>CON</td>
<td>L</td>
<td>LIN1</td>
<td>LIN2</td>
</tr>
<tr>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------</td>
<td>-----</td>
<td>-----</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>30</td>
<td>-20.0000 GY</td>
<td>2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-20.0000 GY</td>
<td>2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-5.0000 GY</td>
<td>15.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

LOADING 9

A SIMPLE BRIDGE DECK

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-10.0000 GY</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-5.0000 GY</td>
<td>25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-5.0000 GY</td>
<td>25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-5.0000 GY</td>
<td>25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-5.0000 GY</td>
<td>25.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

LOADING 10

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>-10.0000 GY</td>
<td>20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-10.0000 GY</td>
<td>20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-10.0000 GY</td>
<td>20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-10.0000 GY</td>
<td>20.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>41</td>
<td>-20.0000 GY</td>
<td>2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>43</td>
<td>-20.0000 GY</td>
<td>2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-5.0000 GY</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-5.0000 GY</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>38</td>
<td>-5.0000 GY</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>37</td>
<td>-5.0000 GY</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

LOADING 11

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>41</td>
<td>-20.0000 GY</td>
<td>2.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### MAXFORCE ENVELOP LIST

#### A SIMPLE BRIDGE DECK

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>FY/</th>
<th>DIST</th>
<th>LD</th>
<th>MZ/</th>
<th>DIST</th>
<th>LD</th>
<th>FX</th>
<th>DIST</th>
<th>LD</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 MAX</td>
<td>18.03</td>
<td>0.00</td>
<td>3</td>
<td>0.02</td>
<td>0.00</td>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td>MIN</td>
<td>-6.97</td>
<td>27.50</td>
<td>3</td>
<td>-373.90</td>
<td>30.00</td>
<td>5</td>
<td>0.00</td>
<td>30.00</td>
<td>11</td>
</tr>
<tr>
<td>41 MAX</td>
<td>16.33</td>
<td>0.00</td>
<td>10</td>
<td>6.80</td>
<td>5.00</td>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td>MIN</td>
<td>-4.08</td>
<td>5.00</td>
<td>11</td>
<td>-109.08</td>
<td>2.50</td>
<td>10</td>
<td>0.00</td>
<td>5.00</td>
<td>11</td>
</tr>
<tr>
<td>42 MAX</td>
<td>0.06</td>
<td>5.00</td>
<td>1</td>
<td>6.80</td>
<td>0.00</td>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>1</td>
</tr>
<tr>
<td>MIN</td>
<td>-0.06</td>
<td>5.00</td>
<td>1</td>
<td>-99.89</td>
<td>5.00</td>
<td>10</td>
<td>0.00</td>
<td>5.00</td>
<td>11</td>
</tr>
</tbody>
</table>

********** END OF FORCE ENVELOPE FROM INTERNAL STORAGE **********

#### FINISH

*********** END OF THE STAAD.Pro RUN ***********

---

**Related Links**

- [TR.31.1 Definition of Moving Load System](on page 2541)
- [V. Moving Load Generator](on page 3546)

**EX. US-13 Section Displacements for a Frame**

Calculation of displacements at intermediate points of members of a plane frame is demonstrated in this example.
This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-13 Section Displacements for a Frame.STD when you install the program.

![Figure 467: Example Problem No. 13](image)

Where:

- \( L_1 = 15 \text{ ft} \)
- \( L_2 = 20 \text{ ft} \)
- \( P = 5.0 \text{ k} \)
- \( w = 3.0 \text{ k/ft} \)

The dashed line represents the deflected shape of the structure. The shape is generated on the basis of displacements at the ends plus several intermediate points of the members.

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE TEST FOR SECTION DISPLACEMENT
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT KIP FEET
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.
Defines the members by the joints to which they are connected.

```
MEMBER PROPERTY AMERICAN
  1 3  TABLE  ST  W8X18
  2  TABLE  ST  W12X26
```

Properties for all members are assigned from the American AISC steel table. The word ST stands for standard single section.

```
UNIT INCHES
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 29000.0
  POISSON 0.3
  DENSITY 283e-006
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
MATERIAL STEEL ALL
```

The input units are changed from FT to INCH. The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
SUPPORT
  1 FIXED ; 4 PINNED
```

A fixed support is specified at Joint 1 and a pinned support at Joint 4.

```
UNIT FT
LOADING 1 DEAD + LIVE + WIND
```

Load case 1 is initiated followed by an optional title.

```
JOINT LOAD
  2 FX 5.
```

Load 1 contains a joint load of 5 kips at node 2. FX indicates that the load is a force in the global X direction.

```
MEMBER LOAD
  2 UNI GY -3.0
```

Load 1 contains a member load also. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
PRINT MEMBER FORCES
```

The above PRINT command is self-explanatory.

```
* FOLLOWING PRINT COMMAND WILL PRINT
* DISPLACEMENTS OF THE MEMBERS
* CONSIDERING EVERY TWELFTH INTERMEDIATE
* POINT (THAT IS TOTAL OF 13 POINTS). THESE
* DISPLACEMENTS ARE MEASURED IN GLOBAL X
* Y Z COORDINATE SYSTEM AND THE VALUES
* ARE FROM ORIGINAL COORDINATES (UNDEФLECTED
* POSITION) OF CORRESPONDING TWELFTH
```
The location of maximum intermediate displacement is determined. This value is measured from above location to the straight line joining start and end joints of the deflected member.

PRINT SECTION DISPLACEMENT

Above PRINT command is explained in the comment lines above.

FINISH

This command terminates the STAAD run.

Input File

STAAD PLANE TEST FOR SECTION DISPLACEMENT
UNIT KIP FEET
JOINT COORDINATES
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W8X18
2 TABLE ST W12X26
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.0
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 FIXED ; 4 PINNED
UNIT FT
LOADING 1 DEAD + LIVE + WIND
JOINT LOAD
2 FX 5.
MEMBER LOAD
2 UNI GY -3.0
PERFORM ANALYSIS
PRINT MEMBER FORCES
*
* FOLLOWING PRINT COMMAND WILL PRINT DISPLACEMENTS
* OF THE MEMBERS CONSIDERING EVERY TWELVETH INTERMEDIATE
* POINTS (THAT IS TOTAL 13 POINTS). THESE DISPLACEMENTS
* ARE MEASURED IN GLOBAL X Y Z COORDINATE SYSTEM AND
* THE VALUES ARE FROM ORIGINAL COORDINATES (THAT IS
* UNDEFORMED) OF CORRESPONDING TWELVETH POINTS.
*
1. STAAD PLANE TEST FOR SECTION DISPLACEMENT

**INPUT FILE:** US-13 Section Displacements for a Frame.STD

2. UNIT KIP FEET

3. JOINT COORDINATES

4. 1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.

5. MEMBER INCIDENCE

6. 1 1 2 ; 2 2 3 ; 3 3 4

7. MEMBER PROPERTY AMERICAN

8. 1 3 TABLE ST W8X18

9. 2 TABLE ST W12X26

10. UNIT INCHES

11. DEFINE MATERIAL START

12. ISOTROPIC STEEL

13. E 29000.0

14. POISSON 0.3

15. DENSITY 283E-006

16. ALPHA 6E-006

17. DAMP 0.03

18. TYPE STEEL

19. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2

20. END DEFINE MATERIAL

21. CONSTANT

22. MATERIAL STEEL ALL

23. SUPPORT

24. 1 FIXED ; 4 PINNED

25. UNIT FT

26. LOADING 1 DEAD + LIVE + WIND

27. JOINT LOAD

28. 2 FX 5.

29. MEMBER LOAD

30. 2 UNI GY -3.0

31. PERFORM ANALYSIS

**TEST FOR SECTION DISPLACEMENT**

**PROBLEM STATISTICS**

| NUMBER OF JOINTS | 4 | NUMBER OF MEMBERS | 3 |
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0  
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2  
Using 64-bit analysis engine.  
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER  
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       7  
TOTAL LOAD COMBINATION CASES =     0  SO FAR.  

32. PRINT MEMBER FORCES

MEMBER FORCES
TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    3
MEMBER END FORCES  STRUCTURE TYPE = PLANE

-----------------
ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
1    1     1     27.37      0.97     0.00      0.00      0.00      22.49
2    -27.37     -0.97     0.00      0.00      0.00      -7.88
2    1     2      4.03     27.37     0.00      0.00      0.00       7.88
3     -4.03     32.63     0.00      0.00      0.00     -60.39
3    1     3     32.63      4.03     0.00      0.00      0.00      60.39
4    -32.63     -4.03     0.00      0.00      0.00       0.00

************** END OF LATEST ANALYSIS RESULT **************

33. *
34. * FOLLOWING PRINT COMMAND WILL PRINT DISPLACEMENTS
35. * OF THE MEMBERS CONSIDERING EVERY TWELVE INTERMEDIATE
36. * POINTS (THAT IS TOTAL 13 POINTS). THESE DISPLACEMENTS
37. * ARE MEASURED IN GLOBAL X Y Z COORDINATE SYSTEM AND
38. * THE VALUES ARE FROM ORIGINAL COORDINATES (THAT IS
39. * UNDEFORMED) OF CORRESPONDING TWELVE POINTS.
40. *
41. * MAX LOCAL DISPLACEMENT IS ALSO PRINTED. THE LOCATION
42. * OF THE MAXIMUM INTERMEDIATE DISPLACEMENT IS DETERMINED.
43. * THIS VALUE IS MEASURED FROM ABOVE LOCATION TO THE STRAIGHT
44. * LINE JOINING START AND END JOINTS OF THE DEFLECTED MEMBER.
45. *

46. PRINT SECTION DISPLACEMENT

SECTION DISPLACE
TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    4
MEMBER SECTION DISPLACEMENTS

-----------------
UNITS ARE - INCH
MEMB  LOAD   GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
1     1       0.0000    0.0000    0.0000    0.0173   -0.0027    0.0000
0.0666   -0.0054    0.0000    0.1461   -0.0081    0.0000
0.2539   -0.0108    0.0000    0.3882   -0.0135    0.0000
0.5472   -0.0162    0.0000    0.7290   -0.0188    0.0000
0.9318   -0.0215    0.0000    1.1538   -0.0242    0.0000
1.3931   -0.0269    0.0000    1.6480   -0.0296    0.0000
1.9165   -0.0323    0.0000
MAX LOCAL  DISP =    0.41112   AT      90.00  LOAD    1   L/DISP=    437
2     1       1.9165   -0.0323    0.0000    1.9162   -0.3903    0.0000
1.9158   -0.7221    0.0000    1.9154   -1.0010    0.0000
1.9151   -1.2067    0.0000    1.9147   -1.3260    0.0000
1.9144   -1.3523    0.0000    1.9140   -1.2856    0.0000
1.9136   -1.1331    0.0000    1.9133   -0.9082    0.0000
1.9129   -0.6316    0.0000    1.9125   -0.3303    0.0000
1.9122   -0.0385    0.0000
MAX LOCAL  DISP =    1.31688   AT     120.00  LOAD    1   L/DISP=    182
3     1       1.9122   -0.0385    0.0000    2.0720   -0.0353    0.0000
2.1486   -0.0321    0.0000    2.1495   -0.0289    0.0000
EX. US-14 P-Delta Analysis of a Frame Under Seismic Loads

A space frame is analyzed for seismic loads. The seismic loads are generated using the procedures of the building code. A P-Delta analysis is performed to obtain the secondary effects of the lateral and vertical loads acting simultaneously.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-14 P-Delta Analysis of a Frame Under Seismic Loads.STD when you install the program.
Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame.

UNIT FEET KIP

Defines the input units for the data that follows.

J O I N T  C O O R D I N A T E S
1 0 0 0 4 3 0 0 0
REPEAT 3 0 0 1 0
REPEAT ALL 3 0 1 0 0

The X, Y and Z coordinates of the joints are specified here. First, coordinates of joints 1 through 4 are generated by taking advantage of the fact that they are equally spaced. Then, this pattern is REPEATED 3 times with a Z increment of 3.5 m for each repetition to generate joints 5 to 16. The REPEAT ALL command will then repeat 3 times, the pattern of joints 1 to 16 to generate joints 17 to 64.
**Application Examples**

**EX. American Design Examples**

```
110 29 30 112
REPEAT ALL 2 12 16
* beams in z direction
201 17 21 204
205 21 25 208
209 25 29 212
REPEAT ALL 2 12 16
* columns
301 1 17 348
```

Defines the members by the joints to which they are connected. Following the specification of incidences for members 101 to 112, the REPEAT ALL command is used to repeat the pattern and generate incidences for members 113 through 136. A similar logic is used in specification of incidences of members 201 through 212 and generation of incidences for members 213 to 236. Finally, members incidences of columns 301 to 348 are specified.

```
UNIT INCH
MEMBER PROPERTIES AMERICAN
101 TO 136 201 TO 236 PRIS YD 15 ZD 15
301 TO 348 TA ST W18X35
```

The beam members have prismatic member property specification (YD & ZD) while the columns (members 301 to 348) have their properties called from the built-in American (AISC) steel table.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL MEMB 301 TO 348
MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

**Tip:** You may see these values with the help of the command PRINT MATERIAL PROPERTIES following the preceding commands.

```
SUPPORT
1 TO 16 FIXED
```
Indicates the joints where the supports are located as well as the type of support restraints.

<table>
<thead>
<tr>
<th>UNIT FEET</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE IBC 2012 LOAD</td>
</tr>
<tr>
<td>ZIP 92806 RX 9 RZ 9 I 1.0 TL 12.0 SCLASS 4 CT 0.032</td>
</tr>
<tr>
<td>SELFWEIGHT</td>
</tr>
<tr>
<td>JOINT WEIGHT</td>
</tr>
<tr>
<td>17 TO 48 WEIGHT 2.5</td>
</tr>
<tr>
<td>49 TO 64 WEIGHT 1.25</td>
</tr>
</tbody>
</table>

There are two stages in a static seismic load. The first stage is to define the code-specified load parameters along with the vertical loads (weights) from which the base shear will be calculated. The vertical loads may be specified in the form of selfweight, joint weights and/or member weights. Member weights are not shown in this example. It is important to note that these vertical loads are used purely in the determination of the horizontal base shear only. In other words, the structure is not analyzed for these vertical loads.

LOAD 1
- IBC LOAD X 0.75
- SELFWEIGHT Y -1.0
- JOINT LOADS
  - 17 TO 48 FY -2.5
  - 49 TO 64 FY -1.25

This is the second stage in which the static seismic load is applied with the help of a load case number, corresponding direction (X in the above case) and a factor by which the generated horizontal loads should be multiplied. Along with the seismic lateral load, deadweight is also added to the same load case. Since we will be doing second-order (PDELTA) analysis, it is important that we include horizontal and vertical loads in the same load case.

LOAD 2
- IBC LOAD Z 0.75
- SELFWEIGHT Y -1.0
- JOINT LOADS
  - 17 TO 48 FY -2.5
  - 49 TO 64 FY -1.25

In load case 2, the static seismic load is being applied in the Z direction. Vertical loads are part of this case, also.

**PDELTA ANALYSIS PRINT LOAD DATA**

We are requesting a second-order analysis by specifying the command **PDELTA ANALYSIS PRINT LOAD DATA** is used to obtain a report of all the applied and generated loadings.

- PRINT SUPPORT REACTIONS
- FINISH

The above commands are self-explanatory.

**Input File**

```
STAAD SPACE EXAMPLE PROBLEM FOR IBC LOAD
UNIT FEET KIP
JOINT COORDINATES
1 0 0 4 30 0 0
REPEAT 3 0 0 10
REPEAT ALL 3 0 10 0
MEMBER INCIDENCES
* beams in x direction
101 17 18 103
104 21 22 106
```
107 25 26 109
110 29 30 112
REPEAT ALL 2 12 16
* beams in z direction
201 17 21 204
205 21 25 208
209 25 29 212
REPEAT ALL 2 12 16
* columns
301 1 17 348
UNIT INCH
MEMBER PROPERTIES AMERICAN
101 TO 136 201 TO 236 PRIS YD 15 ZD 15
301 TO 348 TA ST W18X35
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL MEMB 301 TO 348
MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
SUPPORT
1 TO 16 FIXED
UNIT FEET
DEFINE IBC 2012 LOAD
ZIP 92806 RX 9 RZ 9 I 1.0 TL 12.0 SCLASS 4 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 2.5
49 TO 64 WEIGHT 1.25
LOAD 1
IBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
49 TO 64 FY -1.25
LOAD 2
IBC LOAD Z 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
49 TO 64 FY -1.25
PDELTA ANALYSIS PRINT LOAD DATA
1. STAAD SPACE EXAMPLE PROBLEM FOR IBC LOAD

INPUT FILE: US-14 P-Delta Analysis of a Frame Under Seismic Loads.STD

2. UNIT FEET KIP

3. JOINT COORDINATES
4. 1 0 0 0 4 30 0 0
5. REPEAT 3 0 0 10
6. REPEAT ALL 3 0 10 0

7. MEMBER INCIDENCES
8. * BEAMS IN X DIRECTION
9. 101 17 18 103
10. 104 21 22 106
11. 107 25 26 109
12. 110 29 30 112
13. REPEAT ALL 2 12 16
14. * BEAMS IN Z DIRECTION
15. 201 17 21 204
16. 205 21 25 208
17. 209 25 29 212
18. REPEAT ALL 2 12 16
19. * COLUMNS
20. 301 1 17 348
21. UNIT INCH

22. MEMBER PROPERTIES AMERICAN
23. 101 TO 136 201 TO 236 PRIS YD 15 ZD 15
24. 301 TO 348 TA ST W18X35

25. DEFINE MATERIAL START
26. ISOTROPIC STEEL
27. E 29000
28. POISSON 0.3
29. DENSITY 283E-006
30. ALPHA 6E-006
31. DAMP 0.03
32. TYPE STEEL
33. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
34. ISOTROPIC CONCRETE
35. E 3150
36. POISSON 0.17
37. DENSITY 8.7E-005
38. ALPHA 5E-006

EXAMPLE PROBLEM FOR IBC LOAD

DAMP 0.05
40. G 1346.15
41. TYPE CONCRETE
42. STRENGTH FCU 4
43. END DEFINE MATERIAL
44. CONSTANT
45. MATERIAL STEEL MEMB 301 TO 348
46. MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
47. SUPPORT
48. 1 TO 16 FIXED
49. UNIT FEET
50. DEFINE IBC 2012 LOAD
51. ZIP 92806 RX 9 RZ 9 I 1.0 TL 12.0 SCLASS 4 CT 0.032

*****************************************************************************
* EQUIV. SEISMIC LOADS AS PER IBC 2012                                      *
* PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD GENERATION                      *
*  SS =  1.546 S1 =  0.587 FA =  1.000 FV =  1.500                          *
*  SDS =  1.031 SD1 =  0.587                                                *
*****************************************************************************
52. SELFWEIGHT
53. JOINT WEIGHT
54. 17 TO 48 WEIGHT 2.5
55. 49 TO 64 WEIGHT 1.25
56. LOAD 1
57. IBC LOAD X 0.75
58. SELFWEIGHT Y -1.0
59. JOINT LOADS
60. 17 TO 48 FY -2.5
61. 49 TO 64 FY -1.25
62. LOAD 2
63. IBC LOAD Z 0.75
64. SELFWEIGHT Y -1.0
65. JOINT LOADS
66. 17 TO 48 FY -2.5
67. 49 TO 64 FY -1.25
68. PDELTA ANALYSIS PRINT LOAD DATA

EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    3

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS         64  NUMBER OF MEMBERS     120
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     288
TOTAL LOAD COMBINATION CASES =     0 SO FAR.
EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    4
LOADING 1

---------------
SELFWEIGHT Y -1.000
ACTUAL WEIGHT OF THE STRUCTURE =  185.918 KIP
JOINT LOAD - UNIT KIP FEET
JOINT FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM-Z
17  0.00  -2.50   0.00   0.00   0.00   0.00
18  0.00  -2.50   0.00   0.00   0.00   0.00
19  0.00  -2.50   0.00   0.00   0.00   0.00
20  0.00  -2.50   0.00   0.00   0.00   0.00
21  0.00  -2.50   0.00   0.00   0.00   0.00
22  0.00  -2.50   0.00   0.00   0.00   0.00
### EXAMPLE PROBLEM FOR IBC LOAD

---

**LOADING 2**

**SELFWEIGHT**

Y: -1.000

**ACTUAL WEIGHT OF THE STRUCTURE** = 185.918 KIP

---

**JOINT LOAD - UNIT KIP FEET**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>18</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>21</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>22</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>23</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>24</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>25</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**Example Problem for IBC Load**

---

**Page No. 5**

---

**Actual Weight of the Structure** = 185.918 KIP

---

**Loading 2**

**Selfweight Y: -1.000**

---

**Joint Load - Unit KIP Feet**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>18</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>21</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>22</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>23</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>24</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>25</td>
<td>0.00</td>
<td>-2.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro 4475 User Manual**
**WARNING: IF THIS UBC/IBC ANALYSIS HAS TENSION/COMPRESSION OR REPEAT LOAD OR RE-ANALYSIS OR SELECT OPTIMIZE, THEN EACH UBC/IBC CASE SHOULD BE FOLLOWED BY PERFORM ANALYSIS & CHANGE.**

* IBC 2012 SEISMIC LOAD ALONG X:
  * CT = 0.032 Cu = 1.400 x = 0.8000
  * Ta = 0.486 T = 0.252 Tuser = 0.000
  * TIME PERIODS:
  * Cs LIMITS : LOWER = 0.045 UPPER = 0.259
  * LOAD FACTOR = 0.750
  * DESIGN BASE SHEAR = 0.750 X 0.115 X 285.92 = 24.56 KIP

* IBC 2012 SEISMIC LOAD ALONG Z:
  * CT = 0.032 Cu = 1.400 x = 0.8000
  * TIME PERIODS:
* Ta =  0.486 T =  0.989 Tuser = 0.000  *
* TIME PERIOD USED (T) =  0.681  *
* Cs LIMITS : LOWER =  0.045 UPPER =  0.096  *
* LOAD FACTOR =  0.750  *
* DESIGN BASE SHEAR =  0.750 X 0.096 X 285.92 = 20.56 KIP  *

*****************************************************
JOINT  LATERAL TORSIONAL  LOAD -   1
   LOAD (KIP )  MOMENT (KIP -FEET) FACTOR -  0.750
-----  -------  ---------  ---------
 17   FX  0.235  MY  0.000       
 18   FX  0.288  MY  0.000       
 19   FX  0.288  MY  0.000       

EXAMPLE PROBLEM FOR IBC LOAD

--- PAGE NO.  7 ---

TOTAL = 4.609  0.000 AT LEVEL 10.000 FEET

---
33   FX  0.470  MY  0.000       
34   FX  0.576  MY  0.000       
35   FX  0.576  MY  0.000       
36   FX  0.470  MY  0.000       
37   FX  0.576  MY  0.000       
38   FX  0.682  MY  0.000       
39   FX  0.682  MY  0.000       
40   FX  0.576  MY  0.000       
41   FX  0.576  MY  0.000       
42   FX  0.682  MY  0.000       
43   FX  0.682  MY  0.000       
44   FX  0.576  MY  0.000       
45   FX  0.470  MY  0.000       
46   FX  0.576  MY  0.000       
47   FX  0.576  MY  0.000       
48   FX  0.470  MY  0.000       

TOTAL = 9.218  0.000 AT LEVEL 20.000 FEET

---
49   FX  0.512  MY  0.000       
50   FX  0.671  MY  0.000       
51   FX  0.671  MY  0.000       
52   FX  0.512  MY  0.000       
53   FX  0.671  MY  0.000       
54   FX  0.830  MY  0.000       
55   FX  0.830  MY  0.000       
56   FX  0.671  MY  0.000       
57   FX  0.671  MY  0.000       
58   FX  0.830  MY  0.000       
59   FX  0.830  MY  0.000

---
<table>
<thead>
<tr>
<th>Joint</th>
<th>Lateral Load (kip)</th>
<th>Torsional Moment (kip-feet)</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>FZ</td>
<td>0.184</td>
<td>MY</td>
</tr>
<tr>
<td>18</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>19</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>20</td>
<td>FZ</td>
<td>0.184</td>
<td>MY</td>
</tr>
<tr>
<td>21</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>22</td>
<td>FZ</td>
<td>0.267</td>
<td>MY</td>
</tr>
<tr>
<td>23</td>
<td>FZ</td>
<td>0.267</td>
<td>MY</td>
</tr>
<tr>
<td>24</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>25</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>26</td>
<td>FZ</td>
<td>0.267</td>
<td>MY</td>
</tr>
<tr>
<td>27</td>
<td>FZ</td>
<td>0.267</td>
<td>MY</td>
</tr>
<tr>
<td>28</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>29</td>
<td>FZ</td>
<td>0.184</td>
<td>MY</td>
</tr>
<tr>
<td>30</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>31</td>
<td>FZ</td>
<td>0.225</td>
<td>MY</td>
</tr>
<tr>
<td>32</td>
<td>FZ</td>
<td>0.184</td>
<td>MY</td>
</tr>
<tr>
<td>33</td>
<td>FZ</td>
<td>0.391</td>
<td>MY</td>
</tr>
<tr>
<td>34</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>35</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>36</td>
<td>FZ</td>
<td>0.391</td>
<td>MY</td>
</tr>
<tr>
<td>37</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>38</td>
<td>FZ</td>
<td>0.568</td>
<td>MY</td>
</tr>
<tr>
<td>39</td>
<td>FZ</td>
<td>0.568</td>
<td>MY</td>
</tr>
<tr>
<td>40</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>41</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>42</td>
<td>FZ</td>
<td>0.568</td>
<td>MY</td>
</tr>
<tr>
<td>43</td>
<td>FZ</td>
<td>0.568</td>
<td>MY</td>
</tr>
<tr>
<td>44</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>45</td>
<td>FZ</td>
<td>0.391</td>
<td>MY</td>
</tr>
<tr>
<td>46</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>47</td>
<td>FZ</td>
<td>0.480</td>
<td>MY</td>
</tr>
<tr>
<td>48</td>
<td>FZ</td>
<td>0.391</td>
<td>MY</td>
</tr>
</tbody>
</table>

TOTAL = 7.677 (At Level 20,000 feet)
--- Example Problem for IBC Load --- Page No. 9

\[ \text{TOTAL} = 9.274 \text{ kips at level 30.000 feet} \]

Adjusting Displacements.

*********** END OF DATA FROM INTERNAL STORAGE ***********

--- Example Problem for IBC Load --- Page No. 10

**Support Reactions**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-1.23</td>
<td>10.47</td>
<td>0.01</td>
<td>0.05</td>
<td>-0.00</td>
<td>8.48</td>
</tr>
<tr>
<td>2</td>
<td>0.11</td>
<td>10.25</td>
<td>-1.28</td>
<td>-6.71</td>
<td>0.00</td>
<td>-0.33</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-1.70</td>
<td>17.50</td>
<td>0.01</td>
<td>0.05</td>
<td>-0.00</td>
<td>9.87</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>13.61</td>
<td>-1.27</td>
<td>-6.70</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-1.71</td>
<td>17.01</td>
<td>0.01</td>
<td>0.05</td>
<td>-0.00</td>
<td>9.92</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>13.61</td>
<td>-1.27</td>
<td>-6.70</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-1.45</td>
<td>17.30</td>
<td>0.01</td>
<td>0.05</td>
<td>-0.00</td>
<td>9.13</td>
</tr>
<tr>
<td>2</td>
<td>0.11</td>
<td>10.25</td>
<td>-1.28</td>
<td>-6.71</td>
<td>0.00</td>
<td>-0.33</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-1.26</td>
<td>15.02</td>
<td>0.00</td>
<td>-0.01</td>
<td>-0.00</td>
<td>8.62</td>
</tr>
<tr>
<td>2</td>
<td>0.11</td>
<td>20.00</td>
<td>-1.30</td>
<td>-6.87</td>
<td>0.00</td>
<td>-0.33</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-1.72</td>
<td>22.10</td>
<td>0.00</td>
<td>-0.01</td>
<td>-0.00</td>
<td>10.03</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>23.36</td>
<td>-1.28</td>
<td>-6.86</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>-1.74</td>
<td>21.61</td>
<td>0.00</td>
<td>-0.01</td>
<td>-0.00</td>
<td>10.08</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>23.36</td>
<td>-1.28</td>
<td>-6.86</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>-1.47</td>
<td>21.95</td>
<td>0.00</td>
<td>-0.01</td>
<td>-0.00</td>
<td>9.28</td>
</tr>
<tr>
<td>2</td>
<td>0.11</td>
<td>20.00</td>
<td>-1.30</td>
<td>-6.87</td>
<td>0.00</td>
<td>-0.33</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>-1.26</td>
<td>15.02</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>8.62</td>
</tr>
<tr>
<td>2</td>
<td>0.11</td>
<td>16.98</td>
<td>-1.31</td>
<td>-6.87</td>
<td>0.00</td>
<td>-0.33</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>-1.72</td>
<td>22.10</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>10.03</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>23.36</td>
<td>-1.28</td>
<td>-6.86</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>-1.74</td>
<td>21.61</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>10.08</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>20.35</td>
<td>-1.29</td>
<td>-6.86</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>-1.47</td>
<td>21.95</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>9.28</td>
</tr>
<tr>
<td>2</td>
<td>0.11</td>
<td>16.98</td>
<td>-1.31</td>
<td>-6.87</td>
<td>0.00</td>
<td>-0.33</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>-1.23</td>
<td>10.47</td>
<td>0.01</td>
<td>-0.05</td>
<td>0.00</td>
<td>8.48</td>
</tr>
<tr>
<td>2</td>
<td>0.11</td>
<td>17.52</td>
<td>-1.28</td>
<td>-6.77</td>
<td>0.00</td>
<td>-0.33</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>-1.78</td>
<td>17.50</td>
<td>0.01</td>
<td>-0.05</td>
<td>0.00</td>
<td>9.87</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>20.89</td>
<td>-1.27</td>
<td>-6.77</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>-1.71</td>
<td>17.01</td>
<td>0.01</td>
<td>-0.05</td>
<td>0.00</td>
<td>9.92</td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>20.89</td>
<td>-1.27</td>
<td>-6.77</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>-1.45</td>
<td>17.30</td>
<td>0.01</td>
<td>-0.05</td>
<td>0.00</td>
<td>9.13</td>
</tr>
<tr>
<td>2</td>
<td>-0.11</td>
<td>17.52</td>
<td>-1.28</td>
<td>-6.77</td>
<td>0.00</td>
<td>0.33</td>
<td></td>
</tr>
</tbody>
</table>

*********** END OF LATEST ANALYSIS RESULT ***********

--- Example Problem for IBC Load --- Page No. 11

**Finish**

*********** END OF THE STAAD.Pro RUN ***********

**** DATE= APR 14, 2019 TIME= 22:55:37 ****

* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/ *
* Details about additional assistance from *
EX. US-15 Wind and Floor Load Generation on a Space Frame

A space frame is analyzed for loads generated using the built-in wind and floor load generation facilities.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-15 Wind and Floor Load Generation on a Space Frame.STD when you install the program.

Figure 469: Example Problem No. 15

STAAD SPACE - WIND AND FLOOR LOAD GENERATION

This is a space frame analysis problem. Every STAAD input has to start with the command STAAD. The SPACE specification is used to denote a space (3D) frame.

UNIT FEET KIP

The UNIT specification is used to specify the length and/or force units to be used.

JOINT COORDINATES
1 0 0 0
2 10 0 0
3 21 0 0
The JOINT COORDINATE specification is used to specify the X, Y, and Z coordinates of the joints. Note that the REPEAT ALL command has been used to generate joints for the two upper stories each with a Y increment of 12 ft.

MEMBER INCIDENCES
* Columns
  1 1 9 16
* Beams in the X direction
  17 9 10 18
  19 12 13
  20 14 15 21
  22 17 18 23
  24 20 21
  25 22 23 26
* Beams in the Z direction
  27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
  32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24

The MEMBER INCIDENCE specification is used for specifying member connectivities.

UNIT INCH
MEMBER PROPERTIES AMERICAN
  1 TO 16 TA ST W21X50
  17 TO 26 TA ST W18X35
  27 TO 36 TA ST W14X90

Properties for all members are specified from the built-in American (AISC) steel table. Three different sections have been used.

DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 29000
  POISSON 0.3
  DENSITY 283e-006
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
  CONSTANT
  MATERIAL STEEL ALL

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

SUPPORT
  1 TO 8 FIXED BUT MX MZ

The supports of the structure are defined through the SUPPORT specification. Here all the supports are FIXED with releases specified in the MX (rotation about global X-axis) and MZ (rotation about global Z-axis) directions.
When a structure has to be analyzed for wind loading, the engineer is confronted with the task of first converting an abstract quantity like wind velocity or wind pressure into concentrated loads at joints, distributed loads on members, or pressure loads on plates. The large number of calculations involved in this conversion can be avoided by making use of STAAD’s wind load generation utility. This utility takes wind pressure at various heights as the input, and converts them to values that can then be used as concentrated forces known as joint loads in specific load cases. The input specification is done in two stages. The first stage is initiated above through the \textit{DEFINE WIND LOAD} command. The basic parameters of the WIND loading are specified here. All values need to be provided in the current UNIT system. Each wind category is identified with a \textit{TYPE} number (an identification mark) which is used later to specify load cases.

In this example, two different wind intensities (0.1 Kips/sq. ft and 0.15 Kips/sq. ft) are specified for two different height zones (0 to 12 ft. and 12 to 24 ft.). The \textit{EXPOSURE} specification is used to mitigate or magnify the effect at specific nodes due to special considerations like openings in the structure. In this case, two different exposure factors are specified. The first \textit{EXPOSURE} specification specifies the exposure factor as 0.9 for all joints within the height range (defined as global Y-range) of 11 ft. - 13 ft. The second \textit{EXPOSURE} specification specifies the exposure factor as 0.85 for joints 17, 20 and 22. In the \textit{EXPOSURE} factor specification, the joints may be specified directly or through a vertical range specification.

\textbf{LOAD 1 WIND LOAD IN X-DIRECTION}  
\textit{WIND LOAD X 1.2 TYPE 1}

This is the second stage of input specification for the wind load generation. The term \textit{WIND LOAD} and the direction term that follows are used to specify the wind loading in a particular lateral direction. In this case, \textit{WIND} loading \textit{TYPE 1}, defined previously, is being applied in the global X-direction with a positive multiplication factor of 1.2.

\textbf{LOAD 2 FLOOR LOAD @ Y = 12 FT AND 24 FT}  
\textit{FLOOR LOAD}  
\textit{YRANGE 11.9 12.1 FLOAD -0.45 XRANGE 0.0 10.0 ZRANGE 0.0 20.0}  
\textit{YRANGE 11.9 12.1 FLOAD -0.25 XRANGE 10.0 21.0 ZRANGE 0.0 20.0}  
\textit{YRANGE 23.9 24.1 FLOAD -0.25}

In load case 2 in this problem, a floor load generation is performed. In a floor load generation, a pressure load (force per unit area) is converted by the program into specific points forces and distributed forces on the members located in that region. The \textit{YRANGE}, \textit{XRANGE}, and \textit{ZRANGE} specifications are used to define the area of the structure on which the pressure is acting. The \textit{FLOAD} specification is used to specify the value of that pressure. All values need to be provided in the current UNIT system. For example, in the first line in the above \textit{FLOOR LOAD} specification, the region is defined as being located within the bounds \textit{YRANGE} of 11.9-12.1 ft, \textit{XRANGE} of 0.0-10.0 ft and \textit{ZRANGE} of 0.0-20.0 ft. The \textit{-0.45} signifies that the pressure is 0.45 Kip/sq. ft in the negative global Y direction.

The program will identify the members lying within the specified region and derive member loads on these members based on two-way load distribution.

\textbf{PERFORM ANALYSIS PRINT LOAD DATA}

We can view the values and position of the generated loads with the help of the \textit{PRINT LOAD DATA} command used above along with the \textit{PERFORM ANALYSIS} command.

\textbf{PRINT SUPPORT REACTION}  
\textbf{FINISH}

Above commands are self-explanatory.
Input File

STAAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0
2 10 0 0
3 21 0 0
4 0 0 10
5 10 0 10
6 0 0 20
7 10 0 20
8 21 0 20
REPEAT ALL 2 0 12 0
MEMBER INCIDENCES
* Columns
1 1 9 16
* Beams in the X direction
17 9 10 18
19 12 13
20 14 15 21
22 17 18 23
24 20 21
25 22 23 26
* Beams in the Z direction
27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
UNIT INCH
MEMBER PROPERTIES AMERICAN
1 TO 16 TA ST W21X50
17 TO 26 TA ST W18X35
27 TO 36 TA ST W14X90
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 TO 8 FIXED BUT MX MZ
UNIT FEET
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.15 HEIGHT 12 24
EXPOSURE 0.90 YRANGE 11 13
EXPOSURE 0.85 JOINT 17 20 22
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
LOAD 2 FLOOR LOAD @ Y = 12 ft AND 24 ft
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.45 XRANGE 0.0 10.0 ZRANGE 0.0 20.0
YRANGE 11.9 12.1 FLOAD -0.25 XRANGE 10.0 21.0 ZRANGE 0.0 20.0
YRANGE 23.9 24.1 FLOAD -0.25
PERFORM ANALYSIS PRINT LOAD DATA
PRINT SUPPORT REACTION
FINISH

**Application Examples**

**EX. American Design Examples**

---

**STAAD Output File**

1. STAAD SPACE
   - INPUT FILE: US-15 Wind and Floor Load Generation on a Space Frame.STD
   - UNIT FEET KIP
   - JOINT COORDINATES
   - 1 0 0 0
   - 2 10 0 0
   - 3 21 0 0
   - 4 0 0 10
   - 5 10 0 10
   - 6 0 0 20
   - 7 10 0 20
   - 8 21 0 20
   - 9. REPEAT ALL 2 0 12 0
   - 10. MEMBER INCIDENCES
   - 11. * COLUMNS
   - 12. * BEAMS IN THE X DIRECTION
   - 13. * BEAMS IN THE Z DIRECTION
   - 14. UNIT INCH
   - 15. MEMBER PROPERTIES AMERICAN
   - 16. DEFINE MATERIAL START
   - 17. ISOTROPIC STEEL
   - 18. E 29000
   - 19. POISSON 0.3
   - 20. DENSITY 283E-006
   - 21. ALPHA 6E-006
   - 22. DAMP 0.03

---
38. TYPE STEEL
   STAAD SPACE

39. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2

40. END DEFINE MATERIAL

41. CONSTANT

42. MATERIAL STEEL ALL

43. SUPPORT

44. 1 TO 8 FIXED BUT MX MZ

45. UNIT FEET

46. DEFINE WIND LOAD

*** NOTE: If any floor diaphragm is present in the model Wind Load definition
should be defined after Floor Diaphragm definition. Otherwise wind
load generation may be unsuccessful during analysis.

47. TYPE 1

48. INTENSITY 0.1 0.15 HEIGHT 12 24

49. EXPOSURE 0.90 YRANGE 11 13

50. EXPOSURE 0.85 JOINT 17 20 22

51. LOAD 1 WIND LOAD IN X-DIRECTION

52. WIND LOAD X 1.2 TYPE 1

53. LOAD 2 FLOOR LOAD @ Y = 12FT AND 24FT

54. FLOOR LOAD

55. YRANGE 11.9 12.1 FLOAD -0.45 XRANGE 0.0 10.0 ZRANGE 0.0 20.0

**NOTE** about Floor/OneWay Loads/Weights.
Please note that depending on the shape of the floor you may
have to break up the FLOOR/ONEWAY LOAD into multiple commands.
For details please refer to Technical Reference Manual
Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.

56. YRANGE 11.9 12.1 FLOAD -0.25 XRANGE 10.0 21.0 ZRANGE 0.0 20.0

57. YRANGE 23.9 24.1 FLOAD -0.25

58. PERFORM ANALYSIS PRINT LOAD DATA

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS         24  NUMBER OF MEMBERS      36
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      8
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     112
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

LOADING     1  WIND LOAD IN X-DIRECTION

---------------------
JOINT LOAD - UNIT KIP  FEET

JOINT  FORCE-X  FORCE-Y  FORCE-Z  MOM-X  MOM-Y  MOM-Z
   1  3.60     0.00     0.00    0.00    0.00    0.00
   4  7.20     0.00     0.00    0.00    0.00    0.00
   6  3.60     0.00     0.00    0.00    0.00    0.00
   9  8.10     0.00     0.00    0.00    0.00    0.00
  12 16.20     0.00     0.00    0.00    0.00    0.00
  14  8.10     0.00     0.00    0.00    0.00    0.00
  17  4.59     0.00     0.00    0.00    0.00    0.00
  20  9.18     0.00     0.00    0.00    0.00    0.00
  22  4.59     0.00     0.00    0.00    0.00    0.00

LOADING     2  FLOOR LOAD @ Y = 12FT AND 24FT

---------------------
MEMBER LOAD - UNIT KIP  FEET

MEMBER  UDL  L1  L2  CON  L  LIN1  LIN2
<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>-0.0879 GY</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.2637 GY</td>
<td>0.97</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.4395 GY</td>
<td>1.58</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.6152 GY</td>
<td>2.20</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.7910 GY</td>
<td>2.82</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.9668 GY</td>
<td>3.45</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.1426 GY</td>
<td>4.07</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.3184 GY</td>
<td>4.69</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.3184 GY</td>
<td>5.31</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.1426 GY</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.9668 GY</td>
<td>6.55</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.7910 GY</td>
<td>7.18</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.6152 GY</td>
<td>7.80</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.4395 GY</td>
<td>8.42</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.2637 GY</td>
<td>9.03</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.0879 GY</td>
<td>9.58</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.0879 GY</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.2637 GY</td>
<td>0.97</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.4395 GY</td>
<td>1.58</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.6152 GY</td>
<td>2.20</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.7910 GY</td>
<td>2.82</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.9668 GY</td>
<td>3.45</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.1426 GY</td>
<td>4.07</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.3184 GY</td>
<td>4.69</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.3184 GY</td>
<td>5.31</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.1426 GY</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.9668 GY</td>
<td>6.55</td>
<td></td>
</tr>
</tbody>
</table>

**STAAD SPACE**

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>29</td>
<td>-0.7910 GY</td>
<td>7.18</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.6152 GY</td>
<td>7.80</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.4395 GY</td>
<td>8.42</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.2637 GY</td>
<td>9.03</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.0879 GY</td>
<td>9.58</td>
<td></td>
</tr>
</tbody>
</table>

---

**STAAD.Pro 4486 User Manual**
<p>| | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>27</td>
<td>-1.1426 GY</td>
<td>5.93</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.9668 GY</td>
<td>6.55</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.7910 GY</td>
<td>7.18</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.6152 GY</td>
<td>7.80</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.4395 GY</td>
<td>8.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.2637 GY</td>
<td>9.03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>-0.0879 GY</td>
<td>9.58</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.0879 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.2637 GY</td>
<td>0.97</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.4395 GY</td>
<td>1.58</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.6152 GY</td>
<td>2.20</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.7910 GY</td>
<td>2.82</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.9668 GY</td>
<td>3.45</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-1.1426 GY</td>
<td>4.07</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-1.3184 GY</td>
<td>4.69</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-1.3184 GY</td>
<td>5.31</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-1.1426 GY</td>
<td>5.93</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.9668 GY</td>
<td>6.55</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.7910 GY</td>
<td>7.18</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.6152 GY</td>
<td>7.80</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.4395 GY</td>
<td>8.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.2637 GY</td>
<td>9.03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.0879 GY</td>
<td>9.58</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.0879 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.2637 GY</td>
<td>0.97</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.4395 GY</td>
<td>1.58</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.6152 GY</td>
<td>2.20</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.7910 GY</td>
<td>2.82</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.9668 GY</td>
<td>3.45</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-1.1426 GY</td>
<td>4.07</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-1.3184 GY</td>
<td>4.69</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-1.3184 GY</td>
<td>5.31</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-1.1426 GY</td>
<td>5.93</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.9668 GY</td>
<td>6.55</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.7910 GY</td>
<td>7.18</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.6152 GY</td>
<td>7.80</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.4395 GY</td>
<td>8.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.2637 GY</td>
<td>9.03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.0879 GY</td>
<td>9.58</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.0879 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.2637 GY</td>
<td>0.97</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.4395 GY</td>
<td>1.58</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.6152 GY</td>
<td>2.20</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.7910 GY</td>
<td>2.82</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.9668 GY</td>
<td>3.45</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-1.1426 GY</td>
<td>4.07</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-1.3184 GY</td>
<td>4.69</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-1.3184 GY</td>
<td>5.31</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-1.1426 GY</td>
<td>5.93</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.9668 GY</td>
<td>6.55</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.7910 GY</td>
<td>7.18</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.6152 GY</td>
<td>7.80</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.4395 GY</td>
<td>8.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.2637 GY</td>
<td>9.03</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.0879 GY</td>
<td>9.58</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-0.0879 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-0.2637 GY</td>
<td>0.97</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD - UNIT KIP FEET</th>
<th>MEMBER</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>28</td>
<td>-0.4395 GY 1.58</td>
<td>28</td>
<td>-0.6152 GY 2.20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-0.7910 GY 2.82</td>
<td>28</td>
<td>-0.9668 GY 3.45</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-1.1426 GY 4.07</td>
<td>28</td>
<td>-1.3184 GY 4.69</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-1.3184 GY 5.31</td>
<td>28</td>
<td>-1.3184 GY 5.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-0.9668 GY 6.55</td>
<td>28</td>
<td>-0.9668 GY 7.18</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-0.7910 GY 7.80</td>
<td>28</td>
<td>-0.6152 GY 8.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-0.2637 GY 9.03</td>
<td>28</td>
<td>-0.0879 GY 9.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>-1.1426 GY 10.54</td>
<td>28</td>
<td>-0.0591 GY 10.54</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-0.4395 GY 1.58</td>
<td>31</td>
<td>-0.6152 GY 2.20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-0.7910 GY 2.82</td>
<td>31</td>
<td>-0.9668 GY 3.45</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-1.1426 GY 4.07</td>
<td>31</td>
<td>-1.3184 GY 4.69</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-1.3184 GY 5.31</td>
<td>31</td>
<td>-1.3184 GY 5.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-0.9668 GY 6.55</td>
<td>31</td>
<td>-0.9668 GY 7.18</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-0.7910 GY 7.80</td>
<td>31</td>
<td>-0.6152 GY 8.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-0.2637 GY 9.03</td>
<td>31</td>
<td>-0.0879 GY 9.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-1.1426 GY 10.54</td>
<td>31</td>
<td>-0.0591 GY 10.54</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>-1.3750 GY 5.50</td>
<td>31</td>
<td>-0.8862 GY 5.16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>14.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

STAAD SPACE

-- PAGE NO. 7
<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>-0.8862 GY</td>
<td>5.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.7681 GY</td>
<td>6.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.6499 GY</td>
<td>7.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.5317 GY</td>
<td>7.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.4136 GY</td>
<td>8.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.2954 GY</td>
<td>9.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.1772 GY</td>
<td>9.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-0.0591 GY</td>
<td>10.54</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.8862 GY</td>
<td>4.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.7681 GY</td>
<td>5.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.6499 GY</td>
<td>6.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.5317 GY</td>
<td>6.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.4136 GY</td>
<td>7.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.2954 GY</td>
<td>8.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.1772 GY</td>
<td>8.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-0.0591 GY</td>
<td>9.54</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>-1.3750 GY</td>
<td>0.00</td>
<td>4.50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.0591 GY</td>
<td>0.46</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.1772 GY</td>
<td>1.07</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.2954 GY</td>
<td>1.74</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.4136 GY</td>
<td>2.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.5317 GY</td>
<td>3.11</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.6499 GY</td>
<td>3.79</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.7681 GY</td>
<td>4.48</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.8862 GY</td>
<td>5.16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-1.3750 GY</td>
<td>5.50</td>
<td>10.00</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Member Load - Unit Kip Feet**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>UDL</th>
<th>CON</th>
<th>L1</th>
<th>L2</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
<td>-0.0488 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.1465 GY</td>
<td>0.97</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.2441 GY</td>
<td>1.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.3418 GY</td>
<td>2.20</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.4395 GY</td>
<td>2.82</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.5371 GY</td>
<td>3.45</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.6348 GY</td>
<td>4.07</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.7324 GY</td>
<td>4.69</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.7324 GY</td>
<td>5.31</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.6348 GY</td>
<td>5.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.5371 GY</td>
<td>6.55</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.4395 GY</td>
<td>7.18</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.3418 GY</td>
<td>7.88</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.2441 GY</td>
<td>8.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.1465 GY</td>
<td>9.03</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.0488 GY</td>
<td>9.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.0488 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.1465 GY</td>
<td>0.97</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.2441 GY</td>
<td>1.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.3418 GY</td>
<td>2.20</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.4395 GY</td>
<td>2.82</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.5371 GY</td>
<td>3.45</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.6348 GY</td>
<td>4.07</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.7324 GY</td>
<td>4.69</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.7324 GY</td>
<td>5.31</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.6348 GY</td>
<td>5.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.5371 GY</td>
<td>6.55</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.4395 GY</td>
<td>7.18</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.3418 GY</td>
<td>7.80</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>STAAD SPACE</td>
<td>-- PAGE NO. 10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------------</td>
<td>---------------</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.7681 GY 4.48</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.8862 GY 5.16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.8862 GY 14.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.7681 GY 15.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.6499 GY 16.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.5317 GY 16.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.4136 GY 17.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.2954 GY 18.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.1772 GY 18.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0591 GY 19.54</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0591 GY 0.46</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.1772 GY 1.07</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.2954 GY 1.74</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.4136 GY 2.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.5317 GY 3.11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.6499 GY 3.79</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.7681 GY 4.48</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.8862 GY 5.16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.8862 GY 5.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.7681 GY 6.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.6499 GY 7.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.5317 GY 7.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.4136 GY 8.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.2954 GY 9.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.1772 GY 9.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>-0.0591 GY 10.54</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.8862 GY 4.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.7681 GY 5.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.6499 GY 6.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.5317 GY 6.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.4136 GY 7.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.2954 GY 8.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.1772 GY 8.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-0.0591 GY 9.54</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>-1.3750 GY 0.00 4.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.0591 GY 0.46</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.1772 GY 1.07</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.2954 GY 1.74</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.4136 GY 2.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.5317 GY 3.11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.6499 GY 3.79</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.7681 GY 4.48</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.8862 GY 5.16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-1.3750 GY 5.50 10.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0488 GY 0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1465 GY 0.97</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.2441 GY 1.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.3418 GY 2.20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.4395 GY 2.82</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.5371 GY 3.45</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.6348 GY 4.07</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.7324 GY 4.69</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.7324 GY 5.31</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.6348 GY 5.93</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.5371 GY 6.55</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.4395 GY 7.18</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. American Design Examples

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>24</td>
<td>-0.3418 GY</td>
<td>7.80</td>
</tr>
<tr>
<td>24</td>
<td>-0.2441 GY</td>
<td>8.42</td>
</tr>
<tr>
<td>24</td>
<td>-0.1465 GY</td>
<td>9.03</td>
</tr>
<tr>
<td>24</td>
<td>-0.0488 GY</td>
<td>9.58</td>
</tr>
<tr>
<td>35</td>
<td>-0.0488 GY</td>
<td>0.42</td>
</tr>
<tr>
<td>35</td>
<td>-0.1465 GY</td>
<td>0.97</td>
</tr>
<tr>
<td>35</td>
<td>-0.2441 GY</td>
<td>1.58</td>
</tr>
<tr>
<td>35</td>
<td>-0.3418 GY</td>
<td>2.20</td>
</tr>
<tr>
<td>35</td>
<td>-0.4395 GY</td>
<td>2.82</td>
</tr>
<tr>
<td>35</td>
<td>-0.5371 GY</td>
<td>3.45</td>
</tr>
<tr>
<td>35</td>
<td>-0.6348 GY</td>
<td>4.07</td>
</tr>
<tr>
<td>35</td>
<td>-0.7324 GY</td>
<td>4.69</td>
</tr>
<tr>
<td>35</td>
<td>-0.7324 GY</td>
<td>5.31</td>
</tr>
<tr>
<td>35</td>
<td>-0.6348 GY</td>
<td>5.93</td>
</tr>
<tr>
<td>35</td>
<td>-0.5371 GY</td>
<td>6.55</td>
</tr>
<tr>
<td>35</td>
<td>-0.4395 GY</td>
<td>7.18</td>
</tr>
<tr>
<td>35</td>
<td>-0.3418 GY</td>
<td>7.80</td>
</tr>
<tr>
<td>35</td>
<td>-0.2441 GY</td>
<td>8.42</td>
</tr>
<tr>
<td>35</td>
<td>-0.1465 GY</td>
<td>9.03</td>
</tr>
<tr>
<td>35</td>
<td>-0.0488 GY</td>
<td>9.58</td>
</tr>
<tr>
<td>25</td>
<td>-0.0488 GY</td>
<td>0.42</td>
</tr>
<tr>
<td>25</td>
<td>-0.1465 GY</td>
<td>0.97</td>
</tr>
<tr>
<td>25</td>
<td>-0.2441 GY</td>
<td>1.58</td>
</tr>
<tr>
<td>25</td>
<td>-0.3418 GY</td>
<td>2.20</td>
</tr>
<tr>
<td>25</td>
<td>-0.4395 GY</td>
<td>2.82</td>
</tr>
<tr>
<td>25</td>
<td>-0.5371 GY</td>
<td>3.45</td>
</tr>
<tr>
<td>25</td>
<td>-0.6348 GY</td>
<td>4.07</td>
</tr>
<tr>
<td>25</td>
<td>-0.7324 GY</td>
<td>4.69</td>
</tr>
<tr>
<td>25</td>
<td>-0.7324 GY</td>
<td>5.31</td>
</tr>
<tr>
<td>25</td>
<td>-0.6348 GY</td>
<td>5.93</td>
</tr>
<tr>
<td>25</td>
<td>-0.5371 GY</td>
<td>6.55</td>
</tr>
<tr>
<td>25</td>
<td>-0.4395 GY</td>
<td>7.18</td>
</tr>
<tr>
<td>25</td>
<td>-0.3418 GY</td>
<td>7.80</td>
</tr>
<tr>
<td>25</td>
<td>-0.2441 GY</td>
<td>8.42</td>
</tr>
<tr>
<td>25</td>
<td>-0.1465 GY</td>
<td>9.03</td>
</tr>
<tr>
<td>25</td>
<td>-0.0488 GY</td>
<td>9.58</td>
</tr>
<tr>
<td>33</td>
<td>-0.0488 GY</td>
<td>0.42</td>
</tr>
<tr>
<td>33</td>
<td>-0.1465 GY</td>
<td>0.97</td>
</tr>
<tr>
<td>33</td>
<td>-0.2441 GY</td>
<td>1.58</td>
</tr>
<tr>
<td>33</td>
<td>-0.3418 GY</td>
<td>2.20</td>
</tr>
<tr>
<td>33</td>
<td>-0.4395 GY</td>
<td>2.82</td>
</tr>
<tr>
<td>33</td>
<td>-0.5371 GY</td>
<td>3.45</td>
</tr>
<tr>
<td>33</td>
<td>-0.6348 GY</td>
<td>4.07</td>
</tr>
<tr>
<td>33</td>
<td>-0.7324 GY</td>
<td>4.69</td>
</tr>
<tr>
<td>33</td>
<td>-0.7324 GY</td>
<td>5.31</td>
</tr>
<tr>
<td>33</td>
<td>-0.6348 GY</td>
<td>5.93</td>
</tr>
<tr>
<td>33</td>
<td>-0.5371 GY</td>
<td>6.55</td>
</tr>
<tr>
<td>33</td>
<td>-0.4395 GY</td>
<td>7.18</td>
</tr>
<tr>
<td>33</td>
<td>-0.3418 GY</td>
<td>7.80</td>
</tr>
<tr>
<td>33</td>
<td>-0.2441 GY</td>
<td>8.42</td>
</tr>
<tr>
<td>33</td>
<td>-0.1465 GY</td>
<td>9.03</td>
</tr>
<tr>
<td>33</td>
<td>-0.0488 GY</td>
<td>9.58</td>
</tr>
</tbody>
</table>

************ END OF DATA FROM INTERNAL STORAGE ************

59. PRINT SUPPORT REACTION

SUPPORT REACTION

STAAD SPACE

-- PAGE NO. 11

SUPPORT REACTIONS -UNIT KIP FEET  STRUCTURE TYPE = SPACE
EX. US-16 Time History Analysis for Forcing Function and Ground Motion

Dynamic Analysis (Time History) is performed for a 3 span beam with concentrated and distributed masses. The structure is subjected to “forcing function” and “ground motion” loading. The maxima of joint displacements, member end forces and support reactions are determined.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-16 Time History Analysis for Forcing Function and Ground Motion.STD when you install the program.
Where:

\[ L_1 = 3.5 \text{ ft} \]

**STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS**

Every input file has to start with the word STAAD. The term PLANE signifies that the structure is a plane frame.

**UNITS FEET KIP**

Specifies the units to be used.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0.0 0.0 0.0</td>
</tr>
<tr>
<td>2 0.0 3.5 0.0</td>
</tr>
<tr>
<td>3 0.0 7.0 0.0</td>
</tr>
<tr>
<td>4 0.0 10.5 0.0</td>
</tr>
</tbody>
</table>

Joint number followed by the X, Y and Z coordinates are specified above.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 3</td>
</tr>
</tbody>
</table>
Incidences of members 1 to 3 are specified above.

```
UNIT INCH
MEMBER PROPERTIES
1 2 3 PRIS AX 3.0 IZ 240.0
```

All the members have PRISMATIC property specification. Since this is a plane frame, Area of cross section (AX) and Moment of Inertia (IZ) about the Z axis are adequate for the analysis.

```
SUPPORTS
1 4 PINNED
```

Pinned supports are located at nodes 1 and 4.

```
DEFINE MATERIAL
ISOTROPIC CONCRETE
E 14000
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -0.0001 0.5 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
TYPE 2 ACCELERATION
0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
ARRIVAL TIMES
0.0
DAMPING 0.075
```

There are two stages in the command specification required for a time history analysis. The first stage is defined above. First the characteristics of the time varying load are provided. The loading type may be a forcing function (vibrating machinery) or ground motion (earthquake). The former is input in the form of time-force pairs while the latter is in the form of time-acceleration pairs. Following this data, all possible arrival times for these loads on the structure as well as the modal damping ratio are specified. In this example, the damping ratio is the same (7.5%) for all modes.

```
UNIT FEET
LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 0.5
```

Load case 1 above is a static load. A uniformly distributed force of 0.5 kip/ft acts along the global X direction on all 3 members.

```
LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOIN LOAD
2 3 FX 2.5
TIME LOAD
```
This is the second stage in the command specification for time history analysis. This involves the application of
the time varying load on the structure. The masses that constitute the mass matrix of the structure are specified
through the selfweight and joint load commands. The program will extract the lumped masses from these
weights. Following that, both the TIME LOAD and GROUND MOTION are applied simultaneously.

**Note:** This example is only for illustration purposes and that it may be unlikely that a TIME LOAD and GROUND
MOTION both act on the structure at the same time.

The Time load command is used to apply the Type 1 force, acting in the global X direction, at arrival time number
1, at nodes 2 and 3. The Ground motion, namely, the Type 2 time history loading, is also in the global X direction
at arrival time 1.

The above command initiates the analysis process.

During the analysis, the program calculates joint displacements for every time step. The absolute maximum
value of the displacement for every joint is then extracted from this joint displacement history. So, the value
printed using the above command is the absolute maximum value for each of the six degrees of freedom at each
node.

The member forces and support reactions too are calculated for every time step. For each degree of freedom, the
maximum value of the member force and support reaction is extracted from these histories and reported in the
output file using the above command.

**Input File**

```
STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
UNITS FEET KIP
JOINT COORDINATES
 1 0.0 0.0 0.0
 2 0.0 3.5 0.0
 3 0.0 7.0 0.0
 4 0.0 10.5 0.0
MEMBER INCIDENCES
 1 1 2 3
UNIT INCH
MEMBER PROPERTIES
 1 2 3 PRIS AX 3.0 IZ 240.0
SUPPORTS
 1 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
  E 14000
  POISSON 0.17
  DENSITY 8.7e-005
```
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
TYPE 2 ACCELERATION
0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
ARRIVAL TIMES
0.0
DAMPING 0.075
UNIT FEET
LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 0.5
LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOINT LOAD
2 3 FX 2.5
TIME LOAD
2 3 FX 1 1
GROUND MOTION X 2 1
PERFORM ANALYSIS
UNIT INCH
PRINT JOINT DISPLACEMENTS
UNIT FEET
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
FINISH

STAAD Output File

******************************************************************************
* * STAAD.Pro CONNECT Edition * *
* * Version 22.01.00.** * *
* * Proprietary Program of * *
* * Bentley Systems, Inc. * *
* * Date= APR 14, 2019 * *
* * Time= 22:55:46 * *
* *
* Licensed to: Bentley Systems Inc *
******************************************************************************
1. STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
INPUT FILE: US-16 Time History Analysis for Forcing Function and Ground Motion.STD
2. UNITS FEET KIP
3. JOINT COORDINATES
4. 1 0.0 0.0 0.0
5. 2 0.0 3.5 0.0
6. 3 0.0 7.0 0.0
7. 4 0.0 10.5 0.0
8. MEMBER INCIDENCES
9. 1 1 2 3
10. UNIT INCH
11. MEMBER PROPERTIES
12. 1 2 3 PRIS AX 3.0 IZ 240.0
13. SUPPORTS
14. 1 4 PINNED
15. DEFINE MATERIAL START
16. ISOTROPIC CONCRETE
17. E 14000
18. POISSON 0.17
19. DENSITY 8.7E-005
20. ALPHA 5E-006
21. DAMP 0.05
22. G 1346.15
23. TYPE CONCRETE
24. STRENGTH FCU 4
25. END DEFINE MATERIAL
26. CONSTANTS
27. MATERIAL CONCRETE ALL
28. DEFINE TIME HISTORY
29. TYPE 1 FORCE
30. 0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
31. TYPE 2 ACCELERATION
32. 0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
33. ARRIVAL TIMES
34. 0.0
35. DAMPING 0.075
36. UNIT FEET
37. LOAD 1 STATIC LOAD
38. MEMBER LOAD
39. -- PAGE NO. 2
40. EXAMPLE FOR TIME HISTORY ANALYSIS
41. 1 2 3 UNI GX 0.5
42. LOAD 2 TIME HISTORY LOAD
43. SELFWEIGHT X 1.0
44. SELFWEIGHT Y 1.0
45. JOINT LOAD
46. 2 3 FX 2.5
47. TIME LOAD
48. 2 3 FX 1 1
49. GROUND MOTION X 2 1
50. PERFORM ANALYSIS

PROBLEM STATISTICS

NUMBER OF JOINTS 4  NUMBER OF MEMBERS 3
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 8
TOTAL LOAD COMBINATION CASES = 0 SO FAR.
***NOTE: MASSES DEFINED UNDER LOAD# 2 WILL FORM
THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
MORE MODES WERE REQUESTED THAN THERE ARE FREE MASSES.
NUMBER OF MODES REQUESTED = 6
NUMBER OF EXISTING MASSES IN THE MODEL = 4
NUMBER OF MODES THAT WILL BE USED = 4

*** EIGENSOLUTION : ADVANCED METHOD ***
### Calculated Frequencies for Load Case 2

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (Cycles/Sec)</th>
<th>Period (Sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>14.559</td>
<td>0.06869</td>
</tr>
<tr>
<td>2</td>
<td>56.387</td>
<td>0.01773</td>
</tr>
<tr>
<td>3</td>
<td>944.536</td>
<td>0.00106</td>
</tr>
<tr>
<td>4</td>
<td>1635.985</td>
<td>0.00061</td>
</tr>
</tbody>
</table>

### Modal Weight (Modal Mass Times g) in Kip Generalized

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.021924E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>5.021924E+00</td>
</tr>
<tr>
<td>2</td>
<td>2.258180E-27</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>5.021924E+00</td>
</tr>
<tr>
<td>3</td>
<td>0.000000E+00</td>
<td>2.192400E-02</td>
<td>0.000000E+00</td>
<td>2.192400E-02</td>
</tr>
<tr>
<td>4</td>
<td>0.000000E+00</td>
<td>3.824466E-33</td>
<td>0.000000E+00</td>
<td>2.192400E-02</td>
</tr>
</tbody>
</table>

### Mass Participation Factors

<table>
<thead>
<tr>
<th>Mode</th>
<th>Mass Participation Factors in Percent</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
</tr>
<tr>
<td>1</td>
<td>100.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Actual Modal Damping Used in Analysis

<table>
<thead>
<tr>
<th>Mode</th>
<th>Damping</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.07500000</td>
</tr>
<tr>
<td>2</td>
<td>0.07500000</td>
</tr>
<tr>
<td>3</td>
<td>0.07500000</td>
</tr>
<tr>
<td>4</td>
<td>0.07500000</td>
</tr>
</tbody>
</table>

### Time Step Used in Time History Analysis = 0.00139 Seconds

### Number of Modes Whose Contribution Is Considered = 2

### WARNING: Number of Modes Limited to a Frequency of 360.0 Due to the DT Value Entered.

### Time Duration of Time History Analysis = 2.500 Seconds

### Number of Time Steps in the Solution Process = 1800

### Joint Displacements

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rotan</th>
<th>Y-Rotan</th>
<th>Z-Rotan</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.00103</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.00075</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>0.03537</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.00050</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.02632</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>-0.00038</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.03537</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00050</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.02632</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00038</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00103</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00075</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************
### EX. US-17 User-Provided Tables

The usage of user-provided steel tables is illustrated in this example for the analysis and design of a plane frame. User-provided tables allow you to specify property data for sections not found in the built-in steel section tables.

This problem is installed with the program by default to `C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-17 User-Provided Tables.STD` when you install the program.
Figure 471: Example Problem No. 17

Where:

\[ L_1 = 15 \text{ ft}, \quad L_2 = 20 \text{ ft} \]
\[ P = 5.0 \text{ k} \]
\[ w = 3.0 \text{ k/ft} \]

Actual input is shown in bold lettering followed by explanation.

**STAAD PLANE EXAMPLE FOR USER TABLE**

Every input file has to start with the command STAAD. The PLANE command is used to designate the structure as a plane frame.

**UNIT FT KIP**
The UNIT command sets the length and force units to be used.

**JOINT COORDINATES**

```
1 0. 0.; 2 30 0; 3 0 20 0 6 30 20 0
7 0 35; 8 30 35; 9 7.5 35; 10 22.5 35.
11 15 35; 12 5.38; 13 25 38; 14 10 41; 15 20 41
16 15 44
```

The above set of data is used to provide joint coordinates for the various joints of the structure. The Cartesian system is being used here. The data consists of the joint number followed by global X and Y coordinates. Note that for a space frame, the Z coordinate(s) need to be provided also.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

**MEMBER INCIDENCES**

```
1 1 3; 2 3 7; 3 2 6; 4 6 8; 5 3 4
6 4 5; 7 5 6; 8 7 12; 9 12 14
10 14 16; 11 15 16; 12 13 15; 13 8 13
14 9 12; 15 9 14; 16 11 14; 17 11 15
18 10 15; 19 10 13; 20 7 9
21 9 11; 22 10 11; 23 8 10
```

The above data set contains the member incidence information or the joint connectivity data for each member. This completes the geometry of the structure.

**START USER TABLE**

This command is utilized to set up a user-provided steel table. All user-provided steel tables must start with this command.

**TABLE 1**

Each table needs an unique numerical identification. The above command starts setting up Table no. 1. Up to twenty tables may be specified per run.

**UNIT INCH**

**WIDE FLANGE**

This command is used to specify the section-type as WIDE FLANGE in this table. Note that several section-types such as WIDE FLANGE, CHANNEL, ANGLE, TEE etc. are available for specification (see section 5.19 of the Technical Reference Manual on page 2446).

```
WFL14X30
  8.85 13.84 .27 6.73 .385 291. 19.6 .38 4.0 4.1
WFL21X62
  18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
WFL14X109
```

The above data set is used to specify the properties of three wide flange sections. The data for each section consists of two parts. In the first line, the section-name is provided. You are allowed to provide any section name within twelve characters. The second line contains the section properties required for the particular section-type. Each section-type requires a certain number of data (area of cross-section, depth, moment of inertias etc.) provided in a certain order. For example, in this case, for wide flanges, ten different properties are required.

**TABLE 2**

**ANGLES**

```
LANG25255
  2.5 2.5 .3125 .489 0 0
LANG40404
  4 4 .25 .795 0 0
```
The above command and data lines set up another user provided table consisting of angle sections.

This command signifies the end of the user provided table data set. All user provided table related input must be terminated with this command.

In the above command lines, the member properties are being assigned from the user provided tables created earlier. The word UPT signifies that the properties are from the user-provided table. This is followed by the table number and then the section name as specified in the use- provided table. The numbers 1 or 2 following the word UPT indicate the table from which section names are fetched.

The above command is used to designate members 14 to 23 as truss members.

The MEMBER RELEASE command is used to release the MZ moment at the start joint of member no. 5.

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

The length unit is reset to feet using this command.

The length unit is reset to feet using this command.

The above command set is used to designate supports. Here, joint 1 is designated as a fixed support and joint 2 is designated as a pinned support.
MEMB LOAD
  8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2

The above command set is used to specify the loadings on the structure. In this case, dead and live loads are provided through load case 1. It consists of selfweight, concentrated loads at joints 4, 5 and 11, and distributed loads on some members.

PERFORM ANALYSIS

This command instructs the program to execute the analysis at this point.

PARAMETER
  
  CODE AISC UNIFIED
  BEAM 1.0 ALL
  NSF 0.85 ALL
  KY 1.2 MEMB 3 4

The above commands are used to specify parameters for steel design.

SELECT MEMBER 3 6 9 19

This command will perform selection of members per the AISC ASD steel design code. For each member, the member selection will be performed from the table that was originally used for the specification of the member property. In this case, the selection will be from the respective user tables from which the properties were initially assigned. It may be noted that properties may be provided (and selection may be performed) from built-in steel tables and user provided tables in the same data file.

FINISH

This command terminates a STAAD run.

Input File

STAAD PLANE EXAMPLE FOR USER TABLE
UNIT FT KIP

JOINT COORDINATES
  1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
  7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
  11 15 35 ; 12 5. 38. ; 13 25 38 ; 14 10 41 ; 15 20 41
  16 15 44

MEMBER INCIDENCES
  1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
  6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
  10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
  14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
  18 10 15 ; 19 10 13 ; 20 7 9
  21 9 11 ; 22 10 11 ; 23 8 10

UNIT INCH

START USER TABLE
TABLE 1
WIDE FLANGE
  WFL14X30
  8.85 13.84 .27 6.73 .385 291. 19.6 .38 4.0 4.1
  WFL21X62
  18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
  WFL14X109

TABLE 2
ANGLES
  LANG25255
**MEMBER PROPERTIES**

1 3 4 UPT 1 WFL14X109
2 UPT 1 WFL14X30 ; 5 6 7 UPT 1 WFL21X62
8 TO 13 UPT 1 WFL14X30
14 TO 23 UPT 2 LANG40404

*MEMBER TRUSS

*14 TO 23
MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
UNIT FT
SUPPORT
1 FIXED ; 2 PINNED
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -15. ; 11 FY -35.
MEMB LOAD
8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
PERFORM ANALYSIS
PARAMETER
CODE AISC UNIFIED
BEAM 1.0 ALL
NSF 0.85 ALL
KY 1.2 MEMB 3 4
SELECT MEMB 3 6 9 19
FINISH

**STAAD Output File**

* *****************************************************
* * STAAD.Pro CONNECT Edition                        *
* * Version 22.01.00.*                                *
* * Proprietary Program of                            *
* * Bentley Systems, Inc.                             *
* * Date= APR 14, 2019                                *
* * Time= 22:55:52                                   *
**Licensed to: Bentley Systems Inc**

1. STAAD PLAN EXAMPLE FOR USER TABLE
INPUT FILE: US-17 User-Provided Tables.STD
2. UNIT FT KIP
3. JOINT COORDINATES
4. 1 0 0 ; 2 30 0 ; 3 0 20 0 6 30 20 0
5. 7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
6. 11 15 35 ; 12 5. 38. ; 13 25 38 ; 14 10 41 ; 15 20 41
7. 16 15 44
8. MEMBER INCIDENCES
9. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
10. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
11. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
12. 14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
13. 18 10 15 ; 19 10 13 ; 20 7 9
14. 21 9 11 ; 22 10 11 ; 23 8 10
15. UNIT INCH
16. START USER TABLE
17. TABLE 1
18. WIDE FLANGE
19. WFL14X30
20. 8.85 13.84 .27 6.73 .385 291. 19.6 .38 4.0 4.1
21. WFL21X62
22. 18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
23. WFL14X109
25. TABLE 2
26. ANGLES
27. LANG25255
28. 2.5 2.5 .3125 .489 0 0
29. LANG40404
30. 4 4 .25 .795 0 0
31. END
32. MEMBER PROPERTIES
33. 1 3 4 UPT 1 WFL14X109
34. 2 UPT 1 WFL14X30 ; 5 6 7 UPT 1 WFL21X62
35. 8 TO 13 UPT 1 WFL14X30
36. 14 TO 23 UPT 2 LANG40404
37. *MEMBER TRUSS
38. *14 TO 23
39. EXAMPLE FOR USER TABLE
40. MEMB RELEASE
41. 5 START MZ
42. 14 TO 23 START MPY 0.99 MPZ 0.99
43. UNIT INCH
44. DEFINE MATERIAL START
45. ISOTROPIC STEEL
46. E 29000
47. POISSON 0.3
48. DENSITY 283E-006
49. ALPHA 6E-006
50. DAMP 0.03
51. TYPE STEEL
52. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
53. END DEFINE MATERIAL
54. CONSTANT
55. MATERIAL STEEL ALL
56. BETA 90.0 MEMB 3 4
57. UNIT FT
58. SUPPORT
59. 1 FIXED ; 2 PINNED
60. LOADING 1 DEAD AND LIVE LOAD
61. SELFWEIGHT Y -1.0
62. JOINT LOAD
63. 4 5 FY -15. ; 11 FY -35.
64. MEMB LOAD
65. 8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
66. PERFORM ANALYSIS

PROBLEM STATISTICS

-----------------------------------
NUMBER OF JOINTS         16  NUMBER OF MEMBERS      23
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      43
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

67. PARAMETER
68. CODE AISC UNIFIED
69. BEAM 1.0 ALL
70. NSF 0.85 ALL
71. KY 1.2 MEMB 3 4
72. SELECT MEMB 3 6 9 19

EXAMPLE FOR USER TABLE   -- PAGE NO. 3

STEEL DESIGN
EXAMPLE FOR USER TABLE   -- PAGE NO. 4

STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.1)

*************************************************************************
ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:
=========================================
The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
design results are reported in AISC 360-16 code axis convention.
AISC Spec.        STAAD.Pro        Description
--------------------        ------------------------
X                     Z               Axis typically parallel to the sections principal
major axis.
Y                     Y               Axis typically parallel to the sections principal
minor axis.
Z                     X               Longitudinal axis perpendicular to the cross section.

SECTION FORCES AXIS MAPPING: -
AISC Spec.        STAAD.Pro        Description
--------------------        ------------------------
Pz                     FX              Axial force.
Vy                     FY              Shear force along minor axis.
Vx                     FZ              Shear force along major axis.
Tz                     MX              Torsional moment.
My                     MY              Bending moment about minor axis.
Mx                     MZ              Bending moment about major axis.

*** DESIGN MESSAGES ***:
1. Section classification reported is for the cross section and loadcase that produced the worst case design ratio for flexure/compression Capacity results.
2. Results for any Capacity/Check that is not relevant for a section/loadcase based on the code clause in AISC 360-16 will not be shown in the report.
3. Bending results are reported as being about the relevant axis (X/Y), while the results for shear are reported as being for shear forces along the axis. E.g : Mx indicates bending about the X axis, while Vx indicates shear along the X axis.

*** ABBREVIATIONS ***:

F-T-B = Flexural-Torsional Buckling
L-T-B = Lateral-Torsional Buckling
F-L-B = Flange Local Buckling
W-L-B = Web Local Buckling
L-L-B = Leg Local Buckling
C-F-Y = Compression Flange Yielding
T-F-Y = Tension Flange Yielding

EXAMPLE FOR USER TABLE

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile: ST WFL21X62 (UPT)</th>
<th>Status: PASS</th>
<th>Ratio: 0.908</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>20.00</td>
<td>Eq.H1-1a</td>
<td>Pz: 57.32</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Vy: 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Tz: 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>My: -35.68</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Mx: 0.000</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile: ST WFL21X62 (UPT)</th>
<th>Status: PASS</th>
<th>Ratio: 0.520</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>2.50</td>
<td>Eq.H1-1b</td>
<td>Pz: 3.813</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Vy: 0.1573</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Tz: 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>My: 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Mx: -190.6</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile: ST WFL14X30 (UPT)</th>
<th>Status: PASS</th>
<th>Ratio: 0.504</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>5.83</td>
<td>Eq.H1-3a(H1-1b)</td>
<td>Pz: 37.86</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Vy: 4.144</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Tz: 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>My: 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Mx: -54.45</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Member No:</th>
<th>Profile: ST LANG25255 (UPT)</th>
<th>Status: PASS</th>
<th>Ratio: 0.136</th>
<th>Loadcase: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location:</td>
<td>0.00</td>
<td>Eq.H2-1</td>
<td>Pz: 3.900</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Vy: -.6233E-02</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Tz: 0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>My: -.2453E-03</td>
<td></td>
</tr>
</tbody>
</table>

73. FINISH

**WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
EX. US-18 Stress Calculation for Plate Elements

This is an example which demonstrates the calculation of principal stresses on a finite element.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-18 Stress Calculation for Plate Elements.STD when you install the program.
Figure 472: Example Problem No. 18

Where:

$L = 1 \text{ ft}$

Fixed Supports at Joints 1, 2, 3, 4, 5, 9, 13

Load intensity = 1 psi (pound/in$^2$) in negative global Y direction

Actual input is shown in bold lettering followed by explanation.

**STAAD SPACE SAMPLE CALCULATION FOR**

* ELEMENT STRESSES

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D structure).

**UNIT KIP FEET**

Defines the input units for the data that follows.

**JOINT COORDINATES**

1 0 0 4 3 0 0 0

REPEAT 3 0 0 1

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command is used to generate coordinates of joints 5 to 16 based on the pattern of joints 1 to 4.

**ELEMENT INCIDENCE**

1 1 5 6 2 TO 3

REPEAT 2 3 4
Element connectivities of elements 1 to 3 are defined first, based on which, the connectivities of elements 4 to 9 are generated.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 9 THICK 1.0
```

Elements 1 to 9 have a thickness of 1.0 in.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 4 5 9 13 FIXED
```

Fixed support conditions exist at the above mentioned joints.

```
UNIT POUND
LOAD 1
ELEMENT LOAD
1 TO 9 PRESSURE -1.0
```

A uniform pressure of 1 pound/sq. in is applied on all the elements. In the absence of an explicit direction specification, the load is assumed to act along the local Z axis. The negative value indicates that the load acts opposite to the positive direction of the local Z.

```
PERFORM ANALYSIS
```

The above command instructs the program to proceed with the analysis.

```
PRINT SUPPORT REACTION
```

The above command is self-explanatory.

```
PRINT ELEMENT STRESSES LIST 4
```

Element stresses at the centroid of the element are printed using the above command. The output includes membrane stresses, shear stresses, bending moments per unit width and principal stresses.

```
FINISH
```

The STAAD run is terminated.

**Calculation of principal stresses for element 4**

Calculations are presented for the top surface only.

\[ SX = 0.0 \text{ lb/in}^2 \]
SY = 0.0 lb/in²
SXY = 0.0 lb/in²
MX = 16.90 lb in/in
MY = 85.81 lb in/in
MXY = 36.43 lb in/in
S = 1/6t² = 1/6 · 1² = 0.1667 in³ (Section Modulus)

σₓ = SX + MX/S = 0.0 + 16.90/0.1667 = 101.38 psi
σᵧ = SY + MY/S = 0.0 + 85.81/0.1667 = 514.75 psi
τₓᵧ = SXY + MXY/S = 0.0 + 36.43/0.1667 = 218.54 psi

T_max = \sqrt{\frac{(σₓ - σᵧ)²}{4} + \frac{τₓᵧ²}{4}} = \sqrt{\frac{(101.38 - 514.75)²}{4} + 218.54²} = 300.80 psi

S_max = (σₓ + σᵧ)/2 + T_max = (101.38 + 514.75)/2 + 300.80 = 608.87 psi
S_min = (σₓ + σᵧ)/2 - T_max = (101.38 + 514.75)/2 - 300.80 = 7.27 psi

Angle = \frac{1}{2} \tan^{-1}\left(\frac{2τₓᵧ}{σₓ - σᵧ}\right) = \frac{1}{2} \tan^{-1}\left(\frac{2(218.54)}{101.38 - 514.75}\right) = -23.30°

VON = 0.707\sqrt{(S_{max} - S_{min})² + S_{max}² + S_{min}²} = 0.707\sqrt{(608.87 - 7.27)² + 608.87² + 7.27²} = 605.18 psi

**Input File**

STAAD SPACE SAMPLE CALCULATION FOR
* ELEMENT STRESSES
UNIT KIP FEET
JOINT COORDINATES
1 0 0 4 3 0 0
REPEAT 3 0 0 1
ELEMENT INCIDENCE
1 1 5 6 2 TO 3
REPEAT 2 3 4
UNIT INCH
ELEMENT PROPERTIES
1 TO 9 THICK 1.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 4 5 9 13 FIXED
UNIT POUND
LOAD 1
ELEMENT LOAD
1 TO 9 PRESSURE -1.0
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT ELEMENT STRESSES LIST 4
FINISH

STAAD Output File

*****************************************************************************
*                                                *
*           STAAD.Pro CONNECT Edition           *
*           Version 22.01.00.**                  *
*           Proprietary Program of              *
*           Bentley Systems, Inc.               *
*           Date= APR 14, 2019                  *
*           Time= 22:55:57                      *
*                                                *
* Licensed to: Bentley Systems Inc              *
*****************************************************************************

1. STAAD SPACE SAMPLE CALCULATION FOR
INPUT FILE: US-18 Stress Calculation for Plate Elements.STD

Problem Statistics

2. * ELEMENT STRESSES
3. UNIT KIP FEET
4. JOINT COORDINATES
5. 1 0 0 0 4 3 0 0
6. REPEAT 3 0 0 1
7. ELEMENT INCIDENCE
8. 1 1 5 6 2 TO 3
9. REPEAT 2 3 4
10. UNIT INCH
11. ELEMENT PROPERTIES
12. 1 TO 9 THICK 1.0
13. DEFINE MATERIAL START
14. ISO TROPIC CONCRETE
15. E 3150
16. POISSON 0.17
17. DENSITY 8.7E-005
18. ALPHA 5E-006
19. DAMP 0.05
20. G 1346.15
21. TYPE CONCRETE
22. STRENGTH FCU 4
23. END DEFINE MATERIAL
24. CONSTANTS
25. MATERIAL CONCRETE ALL
26. SUPPORT
27. 1 TO 4 5 9 13 FIXED
28. UNIT POUND
29. LOAD 1
30. ELEMENT LOAD
31. 1 TO 9 PRESSURE -1.0
32. PERFORM ANALYSIS

Sample Calculation For

--- PAGE NO. 2

Problem Statistics

-----PAGE NO. 2-----
### SUPPORT REACTION

* ELEMENT STRESSES

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>-9.76</td>
<td>0.00</td>
<td>-12.51</td>
<td>0.00</td>
<td>12.51</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>70.14</td>
<td>0.00</td>
<td>-853.81</td>
<td>0.00</td>
<td>-16.81</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>301.41</td>
<td>0.00</td>
<td>-2821.43</td>
<td>0.00</td>
<td>95.50</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>281.33</td>
<td>0.00</td>
<td>-2127.22</td>
<td>0.00</td>
<td>-769.10</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00</td>
<td>70.14</td>
<td>0.00</td>
<td>16.81</td>
<td>0.00</td>
<td>853.81</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0.00</td>
<td>301.41</td>
<td>0.00</td>
<td>-95.50</td>
<td>0.00</td>
<td>2821.43</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>0.00</td>
<td>281.33</td>
<td>0.00</td>
<td>769.10</td>
<td>0.00</td>
<td>2127.22</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

### ELEMENT STRESSES

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
<th>TRESPAT</th>
<th>TRESPAB</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>1</td>
<td>10.43</td>
<td>-9.94</td>
<td>16.90</td>
<td>85.81</td>
<td>36.43</td>
<td>608.99</td>
<td>608.99</td>
</tr>
</tbody>
</table>

TOP : SMAX= 608.99 SMIN= -7.25 TMAX= 300.87 ANGLE= 66.7
BOTT: SMAX= -7.25 SMIN= -608.99 TMAX= 300.87 ANGLE=-23.3

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

<table>
<thead>
<tr>
<th>PLATE NO.</th>
<th>CASE NO.</th>
<th>MAXIMUM</th>
<th>MINIMUM</th>
<th>MAXIMUM</th>
<th>MAXIMUM</th>
<th>MAXIMUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>1</td>
<td>6.089873E+02</td>
<td>-6.089873E+02</td>
<td>3.008683E+02</td>
<td>6.053945E+02</td>
<td>6.089873E+02</td>
</tr>
</tbody>
</table>

35. FINISH

*********** END OF THE STAAD.Pro RUN ***********
EX. US-19 Inclined Supports

This example demonstrates the usage of inclined supports. The word INCLINED refers to the fact that the restraints at a joint where such a support is specified are along a user-specified axis system instead of along the default directions of the global axis system. STAAD.Pro offers a few different methods for assigning inclined supports, and we examine those in this example.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-19 Inclined Supports.STD when you install the program.

Figure 473: Example Problem No. 19

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE
INPUT WIDTH 79

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y, and Z coordinates.

UNIT METER KN

Defines the input units for the data that follows.

JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3;

Defines the members by the joints to which they are connected.

UNIT MMS KN
MEMBER PROPERTY AMERICAN
4 5 PRIS YD 800
1 TO 3 PRIS YD 750 ZD 500
```
Properties for all members of the model are provided using the PRISMATIC option (abbreviated to PRIS here). YD and ZD stand for depth and width. If ZD is not provided, a circular shape with diameter = YD is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. The values are provided in MMS unit.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.72
POISSON 0.17
DENSITY 2.35615e-008
ALPHA 5e-006
DAMP 0.05
G 9.281
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
```

Material constants like E (modulus of elasticity) and Poisson's ratio are specified following the command CONSTANTS.

```
UNIT METER KN
SUPPORTS
5 INCLINED REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
6 INCLINED REFJT 3 FIXED BUT MX MY MZ KFX 30000
1 PINNED
4 INCLINED 1 0 1 FIXED BUT FX MX MY MZ
```

We assign supports (restraints) at 4 nodes - 5, 6, 1 and 4. For 3 of those, namely, 5, 6 and 4, the node number is followed by the keyword INCLINED, signifying that an INCLINED support is defined there. For the remaining one - node 1 - that keyword is missing. Hence, the support at node 1 is a global direction support.

The most important aspect of inclined supports is their axis system. Each node where an inclined support is defined has its own distinct local X, local Y and local Z axes. In order to define the axis system, we first have to define a datum point. The support node and the datum point together help define the axis system.

Three different methods are shown in the above 3 instances for defining the datum point.

- At node 5, notice the keyword REF followed by the numbers (10,5,10). This means that the datum point associated with node 5 is one which has the global coordinates of (10m, 5m, 10m). Coincidentally, this happens to be node 2.
- At node 6, the keyword REFJT is used followed by the number 3. This means that the datum point for support node 6 is the joint number 3 of the model. The coordinates of the datum point are hence those of node 3, namely, (20m, 5m and 20m).
- At node 4, the word INCLINED is merely followed by 3 numbers (1,0,1). In the absence of the words REF and REFJT, the program sets the datum point to be the following. It takes the coordinates of node 4, which are (30m,5m,30m) and adds to them, the 3 numbers which comes after the word INCLINED. Thus, the datum point becomes (31m, 5m and 31m).

Once the datum point is established, the local axis system is defined as follows. Local X is a straight line (vector) pointing from the support node towards the datum point. Local Z is the vector obtained by the cross product of local X and the global Y axis (unless the SET Z UP command is used in which case one would use global Z instead of global Y and that would yield local Y). Local Y is the vector resulting from the cross product of local Z and local X. The right hand rule must be used when performing these cross products.
Notice the unique nature of these datum points. The one for node 5 tells us that a line connecting nodes 5 to 2 is the local X axis, and is hence along the axis of member 4. By defining a KFX spring at that one, we are saying that the lower end of member 4 can move along its axis like the piston of a car engine. Think of a pile bored into rock with a certain amount of freedom to expand and contract axially.

The same is true for the support at the bottom of member 5. The local X axis of that support is along the axis of member 5. That also happens to be the case for the supported end of member 3. The line going from node 4 to the datum point (31,5,31) happens to be coincident with the axis of the member, or the traffic direction. The expression FIXED BUT FX MX MY MZ for that support indicates that it is free to translate along local X, suggesting that it is an expansion joint - free to expand or contract along the axis of member 3.

Since MX, MY, and MZ are all released at these supports, no moment will be resisted by these supports.

Three load cases followed by the instruction for the type of analysis are specified. The PRINT STATICS CHECK option will instruct the program to produce a report consisting of total applied load versus total reactions from the supports for each primary load case.

By default, support reactions are printed in the global axis directions. The above command is an instruction for such a report.

Just earlier, we saw how to obtain support reactions in the global axis system. What if we need them in the inclined axis system? The “SET INCLINED REACTION” is a switch for that purpose. It tells the program that reactions should be reported in the inclined axis system instead of the global axis system. This has to be followed by the PRINT SUPPORT REACTIONS command.

Member forces are reported in the local axis system of the members. Joint displacements at all joints are reported in the global axis system. Following this, the STAAD run is terminated.

### Input File

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3;
UNIT MMS KN
MEMBER PROPERTY AMERICAN
4 5 PRIS YD 800
1 TO 3 PRIS YD 750 ZD 500
DEFINE MATERIAL START
```

```
Application Examples
EX. American Design Examples

ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER KN
SUPPORTS
5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
1 PINNED
4 INC 1 0 1 FIXED BUT FX MX MY MZ
LOAD 1 DEAD LOAD
SELFWEIGHT Y -1.2
LOAD 2 LIVE LOAD
MEMBER LOAD
1 TO 3 UNI GY -6
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PRINT SUPPORT REACTION
SET INCLINED REACTION
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
PRINT JOINT DISP
FINISH

STAAD Output File

**********************************************************************************************
*                                                                                          *
*          STAAD.Pro CONNECT Edition                                                      *
*          Version 22.01.00.**                                                            *
*          Proprietary Program of                                                          *
*          Bentley Systems, Inc.                                                          *
*          Date= APR 14, 2019                                                              *
*          Time= 22:56: 2                                                                 *
*                                                                                          *
*          Licensed to: Bentley Systems Inc                                               *
**********************************************************************************************
1. STAAD SPACE
INPUT FILE: US-19 Inclined Supports.STD
2. INPUT WIDTH 79
3. UNIT METER KN
4. JOINT COORDINATES
5. 1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25
6. MEMBER INCIDENCES
7. 1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3
8. UNIT MMS KN
9. MEMBER PROPERTY AMERICAN
10. 4 5 PRIS YD 800

STAAD.Pro 4518 User Manual
11. 1 TO 3 PRIS YD 750 ZD 500
12. DEFINE MATERIAL START
13. ISOTROPIC CONCRETE
14. E 21.0
15. POISSON 0.17
16. DENSITY 2.36158E-008
17. ALPHA 5E-006
18. DAMP 0.05
19. G 9.25
20. TYPE CONCRETE
21. STRENGTH FCU 0.0275
22. END DEFINE MATERIAL
23. CONSTANTS
24. MATERIAL CONCRETE ALL
25. UNIT METER KN
26. SUPPORTS
27. 5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
28. 6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
29. 1 PINNED
30. 4 INC 1 0 1 FIXED BUT FX MX MY MZ
31. LOAD 1 DEAD LOAD
32. SELFWEIGHT Y -1.2
33. LOAD 2 LIVE LOAD
34. MEMBER LOAD
35. 1 TO 3 UNI GY -6
36. LOAD COMB 3
37. 1 1.0 2 1.0
38. PERFORM ANALYSIS PRINT STATICS CHECK

**Problem Statistics**

---

<table>
<thead>
<tr>
<th>NUMBER OF JOINTS</th>
<th>6</th>
</tr>
</thead>
<tbody>
<tr>
<td>NUMBER OF MEMBERS</td>
<td>5</td>
</tr>
<tr>
<td>NUMBER OF PLATES</td>
<td>0</td>
</tr>
<tr>
<td>NUMBER OF SOLIDS</td>
<td>0</td>
</tr>
<tr>
<td>NUMBER OF SURFACES</td>
<td>0</td>
</tr>
<tr>
<td>NUMBER OF SUPPORTS</td>
<td>4</td>
</tr>
</tbody>
</table>

Using 64-bit analysis engine.

SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 27
TOTAL LOAD COMBINATION CASES = 1 SO FAR.

**Static Load/Reaction/Equilibrium Summary for Case No. 1**

DEAD LOAD
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(Forces in Non-Global Directions Will Invalidate Results)

\[
\begin{align*}
X &= 0.150000000E+02 \\
Y &= 0.411580006E+01 \\
Z &= 0.150000000E+02
\end{align*}
\]

TOTAL APPLIED LOAD 1
***TOTAL APPLIED LOAD ( KN METE ) SUMMARY (LOADING 1 )

| SUMMATION FORCE-X | 0.00 |
| SUMMATION FORCE-Y | -697.60 |
| SUMMATION FORCE-Z | 0.00 |

SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 10463.94 MY= 0.00 MZ= -10463.94

TOTAL REACTION LOAD 1
***TOTAL REACTION LOAD( KN METE ) SUMMARY (LOADING 1 )

| SUMMATION FORCE-X | -0.00 |
| SUMMATION FORCE-Y | 697.60 |
| SUMMATION FORCE-Z | -0.00 |
### Summation of Moments Around the Origin

- **MX**: -10463.94
- **MY**: 0.00
- **MZ**: 10463.94

### Maximum Displacements (CM/Radians) (Loading 1)

- **Maxima** at node
  - **X**: -8.02030E-01, 5
  - **Y**: -2.50768E+00, 3
  - **Z**: -8.02030E-01, 5
  - **RX**: -2.71938E-03, 4
  - **RY**: 8.02194E-18, 1
  - **RZ**: 2.71938E-03, 4

### Static Load/Reaction/Equilibrium Summary for Case No. 2

- **Total Applied Load**
  - **SUMMATION FORCE-X**: 0.00
  - **SUMMATION FORCE-Y**: -254.56
  - **SUMMATION FORCE-Z**: 0.00
  - **SUMMATION OF MOMENTS AROUND THE ORIGIN**
    - **MX**: 3818.38
    - **MY**: 0.00
    - **MZ**: -3818.38

- **Total Reaction Load**
  - **SUMMATION FORCE-X**: 0.00
  - **SUMMATION FORCE-Y**: 254.56
  - **SUMMATION FORCE-Z**: 0.00
  - **SUMMATION OF MOMENTS AROUND THE ORIGIN**
    - **MX**: -3818.38
    - **MY**: 0.00
    - **MZ**: 3818.38

### Maximum Displacements (CM/Radians) (Loading 2)

- **Maxima** at node
  - **X**: -2.97766E-01, 5
  - **Y**: -9.34280E-01, 3
  - **Z**: -2.97766E-01, 5
  - **RX**: -1.21481E-03, 4
  - **RY**: -3.94177E-18, 4
  - **RZ**: 1.21481E-03, 4

---

### Support Reaction

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>216.27</td>
<td>289.28</td>
<td>216.27</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>86.61</td>
<td>94.78</td>
<td>86.61</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>302.88</td>
<td>384.06</td>
<td>302.88</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>-213.07</td>
<td>287.50</td>
<td>-213.07</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-85.33</td>
<td>94.06</td>
<td>-85.33</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>-298.40</td>
<td>381.56</td>
<td>-298.40</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-3.20</td>
<td>60.34</td>
<td>-3.20</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>-1.28</td>
<td>32.84</td>
<td>-1.28</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>-4.48</td>
<td>93.18</td>
<td>-4.48</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>60.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>32.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**Application Examples**

**EX. American Design Examples**
### 40. Set Inclined Reaction

#### 41. Print Support Reaction

**Support Reaction**

**STAAD SPACE**

**Support Reactions -Unit KN  METE**

**Structure Type = SPACE**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>416.75</td>
<td>59.61</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>154.72</td>
<td>6.67</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>571.47</td>
<td>66.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>412.02</td>
<td>60.77</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>152.83</td>
<td>7.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>564.85</td>
<td>67.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>-3.20</td>
<td>60.34</td>
<td>-3.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>-1.28</td>
<td>32.84</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-4.48</td>
<td>93.18</td>
<td>-4.48</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>60.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.00</td>
<td>32.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0.00</td>
<td>93.36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**END OF LATEST ANALYSIS RESULT**

### 42. Print Member Forces

**Member Forces**

**STAAD SPACE**

**Member End Forces**

**Structure Type = SPACE**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-4.52</td>
<td>60.34</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>1</td>
<td>4.52</td>
<td>89.95</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-209.31</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>2</td>
<td>1.81</td>
<td>32.84</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-135.62</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>3</td>
<td>-6.33</td>
<td>93.18</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-344.93</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>2</td>
<td>301.33</td>
<td>75.98</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>129.19</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>3</td>
<td>-301.33</td>
<td>74.31</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-117.44</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>2</td>
<td>128.67</td>
<td>42.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>77.86</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>1</td>
<td>-128.67</td>
<td>42.09</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-73.16</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>2</td>
<td>422.00</td>
<td>118.73</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>207.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>3</td>
<td>-422.00</td>
<td>116.41</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-190.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>1</td>
<td>0.00</td>
<td>89.82</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>207.54</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>2</td>
<td>0.00</td>
<td>32.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>342.45</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>3</td>
<td>0.00</td>
<td>32.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>342.45</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>4</td>
<td>0.00</td>
<td>93.36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>93.36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>5</td>
<td>416.75</td>
<td>59.61</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>5</td>
<td>-345.52</td>
<td>41.11</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-80.12</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>3</td>
<td>154.72</td>
<td>6.67</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>57.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>5</td>
<td>571.47</td>
<td>66.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>137.88</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>2</td>
<td>500.25</td>
<td>34.44</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>137.88</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>6</td>
<td>564.85</td>
<td>67.90</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>61.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**END OF LATEST ANALYSIS RESULT**

---

**Application Examples**

**EX. American Design Examples**

---

**STAAD.Pro**

**User Manual**
### Related Links

- *[M. To assign an inclined support](on page 817)*
- *[TR.27.2 Inclined Support Specification](on page 2516)*
- *[Create Support dialog](on page 2983)*

### EX. US-20 Generating a Structure in Cylindrical Coordinates

This example generates the geometry of a cylindrical tank structure using the cylindrical coordinate system. The tank lies on its side in this example.
This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-20
Generating a Structure in Cylindrical Coordinates.STD when you install the program.

In this example, a cylindrical tank is modeled using finite elements. The radial direction is in the XY plane and longitudinal direction is along the Z-axis. Hence, the coordinates in the XY plane are generated using the cylindrical coordinate system.

STAAD SPACE
UNIT KIP FEET

The type of structure (space frame) and length and force units for data to follow are specified.

JOINT COORD CYLINDRICAL

The above command instructs the program that the coordinate data that follows is in the cylindrical coordinate system (r,theta,z).

1 10 0 0 8 10 315 0

Joint 1 has an 'r' of 10 feet, theta of 0 degrees and Z of 0 ft. Joint 8 has an 'r' of 10 feet, theta of 315 degrees and Z of 0 ft. The 315 degrees angle is measured counter-clockwise from the +ve direction of the X-axis. Joints 2 to 7 are generated by equal increments of the coordinate values between joints 1 and 8.

REPEAT 2 0 0 8.5

The REPEAT command is used to generate joints 9 through 24 by repeating twice, the pattern of joints 1 to 8 at Z-increments of 8.5 ft for each REPEAT.

PRINT JOINT COORD
The above command is used to produce a report consisting of the coordinates of all the joints in the Cartesian coordinate system. Note that even though the input data was in the cylindrical coordinate system, the output is in the Cartesian coordinate system.

```
ELEMENT INCIDENCES
  1 1 2 10 9 TO 7 1 1
  8 8 1 9 16
REPEAT ALL 1 8 8
```

The above 4 lines identify the element incidences of all 16 elements. Incidences of element 1 is defined as 1 2 10 9. Incidences of element 2 is generated by incrementing the joint numbers of element 1 by 1, incidences of element 3 is generated by incrementing the incidences of element 2 by 1 and so on up to element 7. Incidences of element 8 has been defined above as 8 1 9 16. The REPEAT ALL command states that the pattern of all the elements defined by the previous 2 lines, namely elements 1 to 8, must be repeated once with an element number increment of 8 and a joint number increment of 8 to generate elements 9 through 16.

```
PRINT ELEMENT INFO
```

The above command is self-explanatory.

```
FINISH
```

### Input File

```
STAAD SPACE
UNIT KIP FEET
JOINT COORD CYLINDRICAL
1 10 0 0 8 10 315 0
REPEAT 2 0 0 8.5
PRINT JOINT COORD
ELEMENT INCIDENCES
  1 1 2 10 9 TO 7 1 1
  8 8 1 9 16
REPEAT ALL 1 8 8
PRINT ELEMENT INFO
FINISH
```

### STAAD Output File

```
******************************************************************************
*                                                                         *
*          STAAD.Pro CONNECT Edition                                        *
*          Version 22.01.00.**                                             *
*          Proprietary Program of                                          *
*          Bentley Systems, Inc.                                           *
*          Date= APR 14, 2019                                             *
*          Time= 22:56:13                                                  *
*                                                                         *
* Licensed to: Bentley Systems Inc                                       *
******************************************************************************

1. STAAD SPACE
INPUT FILE: US-20 Generating a Structure in Cylindrical Coordinates.STD
2. UNIT KIP FEET
3. JOINT COORD CYLINDRICAL
4. 1 10 0 0 8 10 315 0
5. REPEAT 2 0 0 8.5
6. PRINT JOINT COORD
JOINT COORD
```

---

**Application Examples**

**EX. American Design Examples**

The above command is used to produce a report consisting of the coordinates of all the joints in the Cartesian coordinate system. Note that even though the input data was in the cylindrical coordinate system, the output is in the Cartesian coordinate system.

```
ELEMENT INCIDENCES
  1 1 2 10 9 TO 7 1 1
  8 8 1 9 16
REPEAT ALL 1 8 8
```

The above 4 lines identify the element incidences of all 16 elements. Incidences of element 1 is defined as 1 2 10 9. Incidences of element 2 is generated by incrementing the joint numbers of element 1 by 1, incidences of element 3 is generated by incrementing the incidences of element 2 by 1 and so on up to element 7. Incidences of element 8 has been defined above as 8 1 9 16. The REPEAT ALL command states that the pattern of all the elements defined by the previous 2 lines, namely elements 1 to 8, must be repeated once with an element number increment of 8 and a joint number increment of 8 to generate elements 9 through 16.

```
PRINT ELEMENT INFO
```

The above command is self-explanatory.

```
FINISH
```

---

**Input File**

```
STAAD SPACE
UNIT KIP FEET
JOINT COORD CYLINDRICAL
1 10 0 0 8 10 315 0
REPEAT 2 0 0 8.5
PRINT JOINT COORD
ELEMENT INCIDENCES
  1 1 2 10 9 TO 7 1 1
  8 8 1 9 16
REPEAT ALL 1 8 8
PRINT ELEMENT INFO
FINISH
```

**STAAD Output File**

```
******************************************************************************
*                                                                         *
*          STAAD.Pro CONNECT Edition                                        *
*          Version 22.01.00.**                                             *
*          Proprietary Program of                                          *
*          Bentley Systems, Inc.                                           *
*          Date= APR 14, 2019                                             *
*          Time= 22:56:13                                                  *
*                                                                         *
* Licensed to: Bentley Systems Inc                                       *
******************************************************************************

1. STAAD SPACE
INPUT FILE: US-20 Generating a Structure in Cylindrical Coordinates.STD
2. UNIT KIP FEET
3. JOINT COORD CYLINDRICAL
4. 1 10 0 0 8 10 315 0
5. REPEAT 2 0 0 8.5
6. PRINT JOINT COORD
JOINT COORD
```
### Joint Coordinates

Coordinates are feet unit

<table>
<thead>
<tr>
<th>Joint</th>
<th>X (Feet)</th>
<th>Y (Feet)</th>
<th>Z (Feet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>7.071</td>
<td>7.071</td>
<td>0.000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>10.000</td>
<td>0.000</td>
</tr>
<tr>
<td>4</td>
<td>-7.071</td>
<td>7.071</td>
<td>0.000</td>
</tr>
<tr>
<td>5</td>
<td>-10.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>6</td>
<td>-7.071</td>
<td>-7.071</td>
<td>0.000</td>
</tr>
<tr>
<td>7</td>
<td>-0.000</td>
<td>-10.000</td>
<td>0.000</td>
</tr>
<tr>
<td>8</td>
<td>7.071</td>
<td>-7.071</td>
<td>0.000</td>
</tr>
<tr>
<td>9</td>
<td>10.000</td>
<td>0.000</td>
<td>8.500</td>
</tr>
<tr>
<td>10</td>
<td>7.071</td>
<td>7.071</td>
<td>8.500</td>
</tr>
<tr>
<td>11</td>
<td>0.000</td>
<td>10.000</td>
<td>8.500</td>
</tr>
<tr>
<td>12</td>
<td>-7.071</td>
<td>7.071</td>
<td>8.500</td>
</tr>
<tr>
<td>13</td>
<td>-10.000</td>
<td>0.000</td>
<td>8.500</td>
</tr>
<tr>
<td>14</td>
<td>-7.071</td>
<td>-7.071</td>
<td>8.500</td>
</tr>
<tr>
<td>15</td>
<td>-0.000</td>
<td>-10.000</td>
<td>8.500</td>
</tr>
<tr>
<td>16</td>
<td>7.071</td>
<td>-7.071</td>
<td>8.500</td>
</tr>
<tr>
<td>17</td>
<td>10.000</td>
<td>0.000</td>
<td>17.000</td>
</tr>
<tr>
<td>18</td>
<td>7.071</td>
<td>7.071</td>
<td>17.000</td>
</tr>
<tr>
<td>19</td>
<td>0.000</td>
<td>10.000</td>
<td>17.000</td>
</tr>
<tr>
<td>20</td>
<td>-7.071</td>
<td>7.071</td>
<td>17.000</td>
</tr>
<tr>
<td>21</td>
<td>-10.000</td>
<td>0.000</td>
<td>17.000</td>
</tr>
<tr>
<td>22</td>
<td>-7.071</td>
<td>-7.071</td>
<td>17.000</td>
</tr>
<tr>
<td>23</td>
<td>-0.000</td>
<td>-10.000</td>
<td>17.000</td>
</tr>
<tr>
<td>24</td>
<td>7.071</td>
<td>-7.071</td>
<td>17.000</td>
</tr>
</tbody>
</table>

************ End of data from internal storage ************

### Element Incidences

1. 1 1 2 10 9 TO 7 1 1
2. 8 8 1 9 16
3. REPEAT ALL 1 8 8
4. PRINT ELEMENT INFO

### Element Information

<table>
<thead>
<tr>
<th>Element No.</th>
<th>Incidences</th>
<th>Thickness (Feet)</th>
<th>Poisson Ratio</th>
<th>E1/E2</th>
<th>G1/G2</th>
<th>Area</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1 2 10 9</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00 0.000000E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.505619E+01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2 3 11 10</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00 0.000000E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.505619E+01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3 4 12 11</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00 0.000000E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.505619E+01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4 5 13 12</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00 0.000000E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.505619E+01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>5 6 14 13</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00 0.000000E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.505619E+01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>6 7 15 14</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00 0.000000E+00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.505619E+01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### EX. US-21 Analysis of a Structure with Tension-Only Members

This example illustrates the modeling of tension-only members using the **MEMBER TENSION** command.

This problem is installed with the program by default to

```
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-21
```

Analysis of a Structure with Tension-Only Members.STD when you install the program.
It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of “active” members (and hence the stiffness matrix) is load case dependent.

![Figure 475: Example Problem No. 21](image)

where:

- $L = 15\text{ ft}$, $H = 10\text{ ft}$
- Load case 1: $P_1 = 10\text{ kips}$ & $P_2 = 15\text{ kips}$
- Load case 2: $P_3 = -10\text{ kips}$ & $P_4 = -15\text{ kips}$

**STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS**

The input data is initiated with the word STAAD. This structure is a PLANE frame.

**UNIT FEET KIP**

Units for the commands to follow are defined above.

**JOINT COORDINATES**

1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
Joint coordinates of joints 1 to 6 are defined above.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 2 5</td>
</tr>
<tr>
<td>6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5</td>
</tr>
</tbody>
</table>

Incidences of members 1 to 10 are defined.

<table>
<thead>
<tr>
<th>MEMBER TENSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TO 9</td>
</tr>
</tbody>
</table>

Members 6 to 9 are defined as tension-only members using the MEMBER TENSION command. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a compressive force, it is disabled from the structure and the analysis is carried out again with the modified structure.

<table>
<thead>
<tr>
<th>MEMBER PROPERTY AMERICAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 10 TA ST W12X26</td>
</tr>
</tbody>
</table>

All members have been assigned a WIDE FLANGE section from the built in American table.

<table>
<thead>
<tr>
<th>UNIT INCH</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td>E 29000</td>
</tr>
<tr>
<td>POISSON 0.3</td>
</tr>
<tr>
<td>DENSITY 283e-006</td>
</tr>
<tr>
<td>ALPHA 6e-006</td>
</tr>
<tr>
<td>DAMP 0.03</td>
</tr>
<tr>
<td>TYPE STEEL</td>
</tr>
<tr>
<td>STRENGTH FY 36 FU 58 RY 1.5 RT 1.2</td>
</tr>
<tr>
<td>END DEFINE MATERIAL</td>
</tr>
<tr>
<td>CONSTANT</td>
</tr>
<tr>
<td>MATERIAL STEEL ALL</td>
</tr>
</tbody>
</table>

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. The length units have been changed from feet to inch to facilitate the input of these values.

<table>
<thead>
<tr>
<th>SUPPORT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 6 PINNED</td>
</tr>
</tbody>
</table>

The supports are defined above.

<table>
<thead>
<tr>
<th>MEMBER TENSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 TO 9</td>
</tr>
</tbody>
</table>

One or more among the members 6 to 9 may have been inactivated in the analysis.

<table>
<thead>
<tr>
<th>LOAD 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD</td>
</tr>
<tr>
<td>2 FX 15</td>
</tr>
<tr>
<td>3 FX 10</td>
</tr>
</tbody>
</table>

Load 1 is defined above and consists of joint loads at joints 2 and 3.

<table>
<thead>
<tr>
<th>LOAD 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOAD</td>
</tr>
<tr>
<td>4 FX -10</td>
</tr>
<tr>
<td>5 FX -15</td>
</tr>
</tbody>
</table>
Load case 2 is described above.

```
LOAD 3
REPEAT LOAD
  1 1.0 2 1.0
```

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously.

```
PRINT ANALYSIS RESULTS
FINI
```

The analysis results are printed and the run terminated.

**Input File**

---

STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
UNIT FEET KIP
SET NL 3
JOINT COORDINATES
  1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
MEMBER INCIDENCES
  1 1 2 5
  6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
MEMBER TENSION
  6 TO 9
MEMBER PROPERTY AMERICAN
  1 TO 10 TA ST W12X26
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
  E 29000.0
  POISSON 0.3
  DENSITY 283e-006
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
  1 PINNED
  6 PINNED
LOAD 1
JOINT LOAD
  2 FX 15
  3 FX 10
PERFORM ANALYSIS
CHANGE
MEMBER TENSION
  6 TO 9
LOAD 2
JOINT LOAD
  4 FX -10
  5 FX -15
PERFORM ANALYSIS
CHANGE
---

**Application Examples**

EX. American Design Examples
1. STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS

INPUT FILE: US-21 Analysis of a Structure with Tension-Only Members.STD

2. UNIT FEET KIP
3. SET NL 3
4. JOINT COORDINATES
   5. 1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
6. MEMBER INCIDENCES
   7. 1 1 2 5
   8. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
9. MEMBER TENSION
10. 6 TO 9
11. MEMBER PROPERTY AMERICAN
12. 1 TO 10 TA ST W12X26
13. UNIT INCH
14. DEFINE MATERIAL START
15. ISOTROPIC STEEL
16. E 29000.0
17. POISSON 0.3
18. DENSITY 283E-006
19. ALPHA 6E-006
20. DAMP 0.03
21. TYPE STEEL
22. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
23. END DEFINE MATERIAL
24. CONSTANT
25. MATERIAL STEEL ALL
26. SUPPORT
27. 1 PINNED
28. 6 PINNED
29. LOAD 1
30. JOINT LOAD
31. 2 FX 15
32. 3 FX 10
Problem Statistics

- Number of Joints: 6
- Number of Members: 10
- Number of Plates: 0
- Number of Surfaces: 0
- Number of Solids: 0
- Number of Supports: 2

Using 64-bit analysis engine.

Solver used is the in-core advanced math solver.

Total primary load cases = 1, total degrees of freedom = 14
Total load combination cases = 0 so far.

** Problem

** Load Case 1 -- Start Iteration No. 2

** Note - Tension/Compression converged after 2 iterations, Case = 1

34. Change
35. Member Tension
36. 6 to 9
37. Load 2
38. Joint Load
39. 4 FX -10
40. 5 FX -15
41. Perform Analysis

** Load Case 2 -- Start Iteration No. 2

** Note - Tension/Compression converged after 2 iterations, Case = 2

42. Change
43. Member Tension
44. 6 to 9
45. Load 3
46. Repeat Load
47. 1 1.0 2 1.0
48. Perform Analysis

** Load Case 3 -- Start Iteration No. 2

** Note - Tension/Compression converged after 2 iterations, Case = 3

49. Change
50. Load List All
51. Print Analysis Results

Analysis Results

Joint Displacement (Inch Radians) Structure Type = Plane

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rotan</th>
<th>Y-Rotan</th>
<th>Z-Rotan</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0062</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0039</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0007</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.06285</td>
<td>0.00373</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0030</td>
</tr>
<tr>
<td>2</td>
<td>-0.04313</td>
<td>-0.01262</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.00028</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00605</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.09724</td>
<td>0.00387</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0018</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.08929</td>
<td>-0.01613</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.00029</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00408</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0002</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.08929</td>
<td>-0.01613</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0029</td>
</tr>
<tr>
<td>2</td>
<td>-0.09724</td>
<td>0.00387</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.00018</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.00408</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0002</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.04313</td>
<td>0.01262</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0028</td>
</tr>
<tr>
<td>2</td>
<td>0.06285</td>
<td>0.00373</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0003</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.00605</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0039</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.00062</td>
<td></td>
</tr>
</tbody>
</table>
## Support Reactions - Unit KIP Inch

Structure Type = Plane

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-24.91</td>
<td>-23.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.09</td>
<td>23.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-0.85</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>-0.85</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

## Member End Forces - Unit KIP Inch

Structure Type = Plane

All units are -- KIP Inch (Local)

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>Joint</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-6.90</td>
<td>0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>-23.33</td>
<td>-0.89</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-10.81</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>5.46</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>-0.24</td>
<td>0.20</td>
<td>0.00</td>
<td>0.00</td>
<td>6.87</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>2</td>
<td>6.49</td>
<td>-0.43</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.74</td>
<td>-25.59</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>3</td>
<td>9.80</td>
<td>-0.24</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.59</td>
<td>-1.39</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>3</td>
<td>9.80</td>
<td>-0.24</td>
<td>0.00</td>
<td>0.00</td>
<td>18.42</td>
<td>-25.59</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>3</td>
<td>10.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>1.39</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>4</td>
<td>4</td>
<td>6.49</td>
<td>0.43</td>
<td>0.00</td>
<td>0.00</td>
<td>25.59</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>4</td>
<td>-6.49</td>
<td>-0.43</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.59</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>5</td>
<td>-0.24</td>
<td>-0.20</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.59</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>5</td>
<td>6.49</td>
<td>0.43</td>
<td>0.00</td>
<td>0.00</td>
<td>-6.07</td>
<td>1.39</td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>6</td>
<td>6.90</td>
<td>0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>-5.46</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>6</td>
<td>0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>18.42</td>
<td></td>
</tr>
</tbody>
</table>

## Support Reactions - Unit KIP Inch

Structure Type = Plane

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-24.91</td>
<td>-23.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>0.09</td>
<td>23.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-0.85</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>-0.85</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

## Member End Forces - Unit KIP Inch

Structure Type = Plane

All units are -- KIP Inch (Local)

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>Joint</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-29.62</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>5.58</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>5.58</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>4</td>
<td>5.58</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>5</td>
<td>5.58</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>6</td>
<td>5.58</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

## Support Reactions - Unit KIP Inch

Structure Type = Plane

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-8.04</td>
<td>8.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>8.04</td>
<td>-8.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-8.04</td>
<td>-8.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

## Member End Forces - Unit KIP Inch

Structure Type = Plane

All units are -- KIP Inch (Local)
**EX. US-22 Time History Analysis for Sinusoidal Loading**

A space frame structure is subjected to a sinusoidal (dynamic) loading. The commands necessary to describe the sine function are demonstrated in this example. Time History analysis is performed on this model.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-22 Time History Analysis for Sinusoidal Loading.STD when you install the program.
STAAD SPACE

*EXAMPLE FOR HARMONIC LOADING GENERATOR

Every STAAD input file has to begin with the word STAAD.

The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

UNIT KIP FEET

The units for the data that follows are specified above.

JOINT COORDINATES

1 0 0 0 ; 2 15 0 0 ; 3 15 0 15 ; 4 0 0 15
5 0 20 0 ; 6 7.5 20 0 ; 7 15 20 0 ; 8 15 20 7.5
9 15 20 15 ; 10 7.5 20 15 ; 11 0 20 15
12 0 20 7.5

The joint number followed by the X, Y and Z coordinates are specified above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCES

1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12

The members are defined by the joints to which they are connected.

UNIT INCH
MEMBER PROPERTIES
1 TO 12 PRIS YD 12 ZD 12
Members 1 to 12 are defined as PRISmatic sections with width and depth values of 12 inches. The UNIT command is specified to change the units for input from FEET to INCHes.

<table>
<thead>
<tr>
<th>SUPPORTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 4 PINNED</td>
</tr>
</tbody>
</table>

Joints 1 to 4 are declared to be pinned-supported.

```
DEFINE MATERIAL START
  ISOTROPIC CONCRETE
  E 3150
  POISSON 0.17
  DENSITY 8.68e-005
  ALPHA 5e-006
  DAMP 0.05
  G 1346.15
  TYPE CONCRETE
  STRENGTH FCU 4
END DEFINE MATERIAL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
DEFINE TIME HISTORY
  TYPE 1 FORCE
  * FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
  FUNCTION SINE
  AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100
  *
  ARRIVAL TIMES
  0.0
  DAMPING 0.075
```

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the sinusoidal loading are provided.

Each data set is individually identified by the number that follows the TYPE command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word FORCE that follows the TYPE 1 command signifies that this data set is for a forcing function. (If you want to specify an earthquake motion, an ACCELERATION may be specified.)

The command FUNCTION SINE indicates that instead of providing the data set as discrete TIME-FORCE pairs, a sinusoidal function, which describes the variation of force with time, is provided.

The parameters of the sine function, such as FREQUENCY, AMPLITUDE, and number of CYCLES of application are then defined. STAAD internally generates discrete TIME-FORCE pairs of data from the sine function in steps of time defined by the default value (see section 5.31.4 of the Technical Reference Manual (on page 2630) for more information). The arrival time value indicates the relative value of time at which the force begins to act upon the structure. The modal damping ratio for all the modes is set to 0.075.

```
LOAD 1 STATIC LOAD CASE
  MEMBER LOAD
  5 6 7 8 9 10 11 12 UNI GY -1.0
```

The above data describe a static load case.

```
LOAD 2 DYNAMIC LOAD CASE
  SELFWEIGHT X 1.0
```
This is the second stage of command specification for time history analysis. The two sets of data specified here are:

a. the weights for generation of the mass matrix
b. the application of the time varying loads on the structure.

The weights (from which the masses for the mass matrix are obtained) are specified in the form of selfweight and joint loads.

Following that, the sinusoidal force is applied using the TIME LOAD command. The forcing function described by the TYPE 1 load is applied on joints 8 and 12 and it starts to act starting at a time defined by the 1st arrival time number.

The above commands are self explanatory. The FINISH command terminates the STAAD run.

Input File

STAAD SPACE EXAMPLE FOR HARMONIC LOADING GENERATOR
UNIT KIP FEET
JOINT COORDINATES
1 0 0 0 ; 2 15 0 0 ; 3 15 0 15 ; 4 0 0 15
5 0 20 0 ; 6 7.5 20 0 ; 7 15 20 0 ; 8 15 20 7.5
9 15 20 15 ; 10 7.5 20 15 ; 11 0 20 15
12 0 20 7.5
MEMBER INCIDENCES
1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
UNIT INCH
MEMBER PROPERTIES
1 TO 12 PRIS YD 12 ZD 12
SUPPORTS
1 TO 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
DEFINE TIME HISTORY
TYPE 1 FORCE
* FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100
*
ARRIVAL TIMES
0.0
DAMPING 0.075
LOAD 1 STATIC LOAD CASE
MEMBER LOAD
5 6 7 8 9 10 11 12 UNI GY -1.0
LOAD 2 DYNAMIC LOAD CASE
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
8 12 FX 4.0
8 12 FY 4.0
8 12 FZ 4.0
TIME LOAD
8 12 FX 1 1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINI

STAAD Output File
19. POISSON 0.17
20. DENSITY 8.68E-005
21. ALPHA 5E-006
22. DAMP 0.05
23. G 1346.15
24. TYPE CONCRETE
25. STRENGTH FCU 4
26. END DEFINE MATERIAL
27. CONSTANTS
28. MATERIAL CONCRETE ALL
29. DEFINE TIME HISTORY
30. TYPE 1 FORCE
31. * FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
32. FUNCTION SINE
33. AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100
   EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 2
34. FOR SEQUENTIAL HARMONIC FORCING CURVE NUMBER=
35. NUMBER OF POINTS IN DIGITIZED HARMONIC FUNCTION=
36. NUMBER OF POINTS PER QUARTER CYCLE OF HARMONIC FUNCTION=
37. FORCE STEP DELTA TIME PER POINT 1.38889E-03
38. ENDING TIME FOR THIS DIGITIZED HARMONIC FUNCTION 1.66667E+00
39. *
40. ARRIVAL TIMES
41. 0.0
42. DAMPING 0.075
43. LOAD 1 STATIC LOAD CASE
44. MEMBER LOAD
45. 5 6 7 8 9 10 11 12 UNI GY -1.0
46. LOAD 2 DYNAMIC LOAD CASE
47. SELFWEIGHT X 1.0
48. SELFWEIGHT Y 1.0
49. SELFWEIGHT Z 1.0
50. JOINT LOAD
51. 8 12 FX 4.0
52. 8 12 FY 4.0
53. 8 12 FZ 4.0
54. TIME LOAD
55. 8 12 FX 1 1
56. PERFORM ANALYSIS

PROBLEM STATISTICS
--------------------------------------------------------
NUMBER OF JOINTS 12  NUMBER OF MEMBERS 12
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 4
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 60
TOTAL LOAD COMBINATION CASES = 0 SO FAR.
***NOTE: MASSES DEFINED UNDER LOAD# 2 WILL FORM
THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 3
EIGEN METHOD : SUBSPACE
------------------------
NUMBER OF MODES REQUESTED = 6
NUMBER OF EXISTING MASSES IN THE MODEL = 24
NUMBER OF MODES THAT WILL BE USED = 6
*** EIGENSOLUTION : ADVANCED METHOD ***
EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 4
52. PRINT ANALYSIS RESULTS

BASE SHEAR UNITS ARE -- KIP  INCH
MAXIMUM BASE SHEAR X=  -2.228746E-01  Y=   1.490116E-08  Z=   2.273737E-13
AT TIMES  0.194444  1.519444  0.072222

ANALYSIS RESULTS
EXAMPLE FOR HARMONIC LOADING GENERATOR                    -- PAGE NO.  6
JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = SPACE

----------------------------------
JOINT LOAD  X-TRANS  Y-TRANS  Z-TRANS  X-ROTAN  Y-ROTAN  Z-ROTAN
1  1   0.00000   0.00000   0.00000  -0.01045   0.00000   0.01045
2  2   0.00000   0.00000   0.00000   0.01045   0.00000  -0.01045
3  2   0.00000   0.00000   0.00000  -0.00000   0.00000  -0.00037
4  3   0.00000   0.00000   0.00000   0.01045   0.00000  -0.01045
5  3   0.00000   0.00000   0.00000   0.00000  -0.00000  -0.00037
6  4   0.00118  -0.09524   0.00118   0.02103   0.00000  -0.02103
7  5   0.06536   0.00008   0.00000  -0.00000   0.00000  -0.00000
8  5   0.00000   0.00000   0.00000   0.00000  -0.00000  -0.00000
9  6   0.00118  -0.09524   0.00118   0.02103   0.00000  -0.02103
10 6   0.06536   0.00008   0.00000  -0.00000   0.00000  -0.00000
11 6   0.00118  -0.09524   0.00118   0.02103   0.00000  -0.02103
12 6   0.06536   0.00008   0.00000  -0.00000   0.00000  -0.00000

CALCULATED FREQUENCIES FOR LOAD CASE  2

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (CYCLES/SEC)</th>
<th>PERIOD (SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.202</td>
<td>0.83191</td>
</tr>
<tr>
<td>2</td>
<td>1.204</td>
<td>0.83057</td>
</tr>
<tr>
<td>3</td>
<td>1.451</td>
<td>0.68908</td>
</tr>
<tr>
<td>4</td>
<td>7.559</td>
<td>0.13229</td>
</tr>
<tr>
<td>5</td>
<td>11.073</td>
<td>0.09031</td>
</tr>
<tr>
<td>6</td>
<td>11.670</td>
<td>0.08569</td>
</tr>
</tbody>
</table>

MODAL WEIGHT (MODAL MASS TIMES g) IN KIP

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.299868E+01</td>
<td>5.214192E-29</td>
<td>2.154110E-25</td>
<td>2.280506E+01</td>
</tr>
<tr>
<td>2</td>
<td>2.148403E-25</td>
<td>9.982997E-31</td>
<td>2.299902E+01</td>
<td>2.293083E+01</td>
</tr>
<tr>
<td>3</td>
<td>2.180342E-21</td>
<td>1.626548E-31</td>
<td>2.281520E-21</td>
<td>3.300665E+01</td>
</tr>
<tr>
<td>4</td>
<td>1.122062E-24</td>
<td>1.446906E-29</td>
<td>2.731749E-24</td>
<td>2.026121E+01</td>
</tr>
<tr>
<td>5</td>
<td>4.06138E-23</td>
<td>5.354261E-06</td>
<td>4.306781E-20</td>
<td>1.127253E+01</td>
</tr>
<tr>
<td>6</td>
<td>1.341889E-20</td>
<td>1.076830E+01</td>
<td>1.183963E-23</td>
<td>1.025517E+01</td>
</tr>
</tbody>
</table>

MASS PARTICIPATION FACTORS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>100.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.998</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>100.00</td>
<td>99.998</td>
<td>100.000</td>
<td>100.000</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.998</td>
<td>0.000</td>
<td>100.000</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.998</td>
<td>0.000</td>
<td>100.000</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.998</td>
<td>0.000</td>
<td>100.000</td>
</tr>
<tr>
<td>6</td>
<td>0.00</td>
<td>46.82</td>
<td>0.00</td>
<td>99.998</td>
<td>46.821</td>
<td>100.000</td>
</tr>
</tbody>
</table>

EXAMPLE FOR HARMONIC LOADING GENERATOR                    -- PAGE NO.  5

ACTUAL MODAL DAMPING USED IN ANALYSIS

<table>
<thead>
<tr>
<th>MODE</th>
<th>DAMPING</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.07500000</td>
</tr>
<tr>
<td>2</td>
<td>0.07500000</td>
</tr>
<tr>
<td>3</td>
<td>0.07500000</td>
</tr>
<tr>
<td>4</td>
<td>0.07500000</td>
</tr>
<tr>
<td>5</td>
<td>0.07500000</td>
</tr>
<tr>
<td>6</td>
<td>0.07500000</td>
</tr>
</tbody>
</table>

TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED = 6
TIME DURATION OF TIME HISTORY ANALYSIS = 1.667 SECONDS
NUMBER OF TIME STEPS IN THE SOLUTION PROCESS = 1200
### Application Examples

**EX. American Design Examples**

#### SUPPORT REACTIONS -UNIT KIP INCH - STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>5.95</td>
<td>180.00</td>
<td>5.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-0.06</td>
<td>-0.15</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-5.95</td>
<td>180.00</td>
<td>5.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-0.06</td>
<td>0.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-5.95</td>
<td>180.00</td>
<td>-5.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-0.06</td>
<td>0.15</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>5.95</td>
<td>180.00</td>
<td>-5.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-0.06</td>
<td>-0.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 7

#### MEMBER END FORCES - STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>MEMBER LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>180.00</td>
<td>-5.95</td>
<td>5.95</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>-0.15</td>
<td>0.06</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>13.37</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>180.00</td>
<td>5.95</td>
<td>5.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>-0.15</td>
<td>-0.06</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>13.37</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>180.00</td>
<td>5.95</td>
<td>-5.95</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td>-180.00</td>
<td>5.95</td>
<td>5.95</td>
<td>0.00</td>
<td>1428.10</td>
<td>1428.10</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.15</td>
<td>-0.06</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td>-0.15</td>
<td>-0.06</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>13.37</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>180.00</td>
<td>-5.95</td>
<td>-5.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>-180.00</td>
<td>5.95</td>
<td>5.95</td>
<td>0.00</td>
<td>1428.10</td>
<td>-1428.10</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>-0.15</td>
<td>0.06</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>0.15</td>
<td>-0.06</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>13.37</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>5.95</td>
<td>90.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>1428.10</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>-5.95</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>2621.90</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>-0.01</td>
<td>-0.15</td>
<td>-0.01</td>
<td>-0.00</td>
<td>0.00</td>
<td>-13.37</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>0.01</td>
<td>0.15</td>
<td>0.01</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>5.95</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-1428.10</td>
</tr>
<tr>
<td>7</td>
<td></td>
<td>-5.95</td>
<td>90.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>2621.90</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>0.01</td>
<td>-0.15</td>
<td>-0.01</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td></td>
<td>-0.01</td>
<td>0.15</td>
<td>0.01</td>
<td>0.00</td>
<td>0.00</td>
<td>-13.37</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>5.95</td>
<td>90.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>1428.10</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td>-5.95</td>
<td>-0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2621.90</td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>-0.01</td>
<td>0.00</td>
<td>0.02</td>
<td>0.00</td>
<td>-0.84</td>
<td>0.00</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. American Design Examples**

```
8  0.01 -0.00 -0.02 -0.00  -1.41  0.00
9 -5.95  90.00  0.00  -0.00  -0.00  -1428.10
10 -0.01  0.00  0.02  -0.00   0.84  -0.00
9  5.95  90.00  0.00  -0.00   0.00  1428.10
12 -0.01  0.00  0.02  -0.00   1.41  -0.00

EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 9
MEMBER END FORCES  STRUCTURE TYPE = SPACE
-----------------------------
ALL UNITS ARE -- KIP INCH  (LOCAL )
MEMBER  LOAD  JT  AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
2     9     0.01     0.15     0.01     0.00     -0.84      13.37
10    -0.01    -0.15    -0.01    -0.00     -0.00     -0.00
11    -5.95    90.00     0.00    -0.00     -0.00     -2621.90
2     10    -0.01    -0.15    -0.01    -0.00     -0.84      13.37
11     0.01    -0.15    -0.01    -0.00     -0.00
12    -5.95    90.00     0.00    -0.00     -0.00     -2621.90
11    -0.01    -0.00    -0.02    -0.00     -0.84      13.37
12     0.01     0.00    -0.02     0.00     -1.41      0.00
5     -0.01     0.00    -0.02     0.00     -0.84      0.00

*************** END OF LATEST ANALYSIS RESULT ***************
53. FINI

*********** END OF THE STAAD.Pro RUN ***********
**** DATE= APR 14,2019   TIME= 22:56:23 ****
```

**EX. US-23 Spring Support Generation for a Slab on Grade**

This example illustrates the usage of commands necessary to automatically generate spring supports for a slab on grade. The slab is subjected to pressure loading and analysis of the structure is performed.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-23
Spring Support Generation for a Slab on Grade.STD when you install the program.
where:

\[
H = 129', h1 = 8' - 6'', h2 = 8', h3 = 6' \\
V = 40', v1 = 6' - 6'', v2 = 6', v3 = 7' - 10'', v4 = 4'
\]

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The remainder of the words form a title to identify this project.

UNIT FEET KIP

The units for the data that follows are specified above.

JOINT COORDINATES
1 0.0 0.0 40.0  
2 0.0 0.0 36.0  
3 0.0 0.0 28.167  
4 0.0 0.0 20.333  
5 0.0 0.0 12.5  
6 0.0 0.0  6.5  
7 0.0 0.0  0.0  
REPEAT ALL 3 8.5 0.0 0.0  
REPEAT 3 8.0 0.0 0.0  
REPEAT 5 6.0 0.0 0.0  
REPEAT 3 8.0 0.0 0.0  
REPEAT 3 8.5 0.0 0.0  

For joints 1 through 7, the joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these seven joints by 8.5 feet, three times REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

ELEMENT INCIDENCES
1 1 8 9 2 TO 6  
REPEAT 16 6 7
The incidences of element number 1 is defined and that data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first six elements is then used to generate the incidences of 96 (= 16 x 6) more elements using the REPEAT command.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 5.5
```

The thickness of elements 1 to 102 is specified as 5.5 inches following the command ELEMENT PROPERTIES.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2916.7
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUB 10.0
```

The above command is used to instruct STAAD to generate supports with springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade modulus of the soil is specified as 10 kip/cu.ft. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the FY degree of freedom at the joint. See Section 5.27.3 in the STAAD Technical Reference Manual (on page 2517).

```
PRINT SUPP INFO
```

This command will enable us to obtain the details of the support conditions which were generated using the earlier commands.

```
LOAD 1 WEIGHT OF MAT & EARTH
ELEMENT LOAD
1 TO 102 PR GY -1.55
```

The above data describe a static load case. A pressure load of 1.55 kip/sq.ft acting in the negative global Y direction is applied on all the 102 elements.

```
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2 FY -217.
8 9 FY -109.
5 FY -308.7
6 FY -617.4
22 23 FY -410.
29 30 FY -205.
26 FY -542.7
27 FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82 FY -264.2
```
Load case 2 consists of several joint loads acting in the negative global Y direction.

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Load Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>08 59 73 80</td>
<td>-528.3</td>
</tr>
<tr>
<td>92 93</td>
<td>-205.0</td>
</tr>
<tr>
<td>99 100</td>
<td>-410.0</td>
</tr>
<tr>
<td>103</td>
<td>-487.0</td>
</tr>
<tr>
<td>104</td>
<td>-974.0</td>
</tr>
<tr>
<td>113 114</td>
<td>-109.0</td>
</tr>
<tr>
<td>120 121</td>
<td>-217.0</td>
</tr>
<tr>
<td>124</td>
<td>-273.3</td>
</tr>
<tr>
<td>125</td>
<td>-546.6</td>
</tr>
</tbody>
</table>

A load combination case, identified with load case number 101, is specified above. It instructs STAAD to factor loads 1 and 2 by a value of 1.0 and then algebraically add the results.

**PERFORM ANALYSIS**

The analysis is initiated using the above command.

**LOAD LIST 101**
**PRINT JOINT DISPLACEMENTS LIST 33 56**
**PRINT ELEMENT STRESSES LIST 34 67**

Joint displacements for joints 33 and 56, and element stresses for elements 34 and 67, for load case 101, is obtained with the help of the above commands.

**FINISH**

The STAAD run is terminated.

**Input File**

```
STAAD SPACE SLAB ON GRADE
UNIT FEET KIP
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0  6.5
7 0.0 0.0  0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 5.5
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2916.7
POISSON 0.12
DENSITY 8.68e-005
```
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 10.0
PRINT SUPP INFO
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.55
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2 FY -217.
8 9 FY -109.
5 FY -308.7
6 FY -617.4
22 23 FY -410.
29 30 FY -205.
26 FY -542.7
27 FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82 FY -264.2
48 55 76 83 FY -528.3
92 93 FY -205.0
99 100 FY -410.0
103 FY -487.0
104 FY -974.0
113 114 FY -109.0
120 121 FY -217.0
124 FY -273.3
125 FY -546.6
LOADING COMBINATION 101 TOTAL LOAD
1 1. 2 1.
PERFORM ANALYSIS
LOAD LIST 101
PRINT JOINT DISPLACEMENTS LIST 33 56
PRINT ELEMENT STRESSES LIST 34 67
FINISH

STAAD Output File

****************************************************************************************** PAGE NO. 1
*  
*  STAAD.Pro CONNECT Edition  
*  Version 22.01.00.**  
*  Proprietary Program of  
*  Bentley Systems, Inc.  
*  Date= APR 14, 2019  
*  Time= 22:56:27  
*  Licensed to: Bentley Systems Inc  
******************************************************************************************
1. STAAD SPACE SLAB ON GRADE
INPUT FILE: US-23 Spring Support Generation for a Slab on Grade.STD
2. UNIT FEET KIP
3. JOINT COORDINATES
4. 1 0.0 0.0 40.0
5. 2 0.0 0.0 36.0
6. 3 0.0 0.0 28.167
7. 4 0.0 0.0 20.333
8. 5 0.0 0.0 12.5
9. 6 0.0 0.0 6.5
10. 7 0.0 0.0 0.0
11. REPEAT ALL 3 8.5 0.0 0.0
12. REPEAT 3 8.0 0.0 0.0
13. REPEAT 5 6.0 0.0 0.0
14. REPEAT 3 8.0 0.0 0.0
15. REPEAT 3 8.5 0.0 0.0
16. ELEMENT INCIDENCES
17. 1 1 8 9 2 TO 6
18. REPEAT 16 6 7
19. UNIT INCH
20. ELEMENT PROPERTIES
21. 1 TO 102 TH 5.5
22. DEFINE MATERIAL START
23. ISOTROPIC CONCRETE
24. E 2916.7
25. POISSON 0.12
26. DENSITY 8.68E-005
27. ALPHA 5E-006
28. DAMP 0.05
29. G 1346.15
30. TYPE CONCRETE
31. STRENGTH FCU 4
32. END DEFINE MATERIAL
33. CONSTANTS
34. MATERIAL CONCRETE ALL
35. UNIT FEET
36. SUPPORTS
37. 1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 10.0
38. PRINT SUPP INFO
SUPPORT INFORMATION (1=FIXED, 0=RELEASED)

UNITS FOR SPRING CONSTANTS ARE KIP FEET DEGREES

<table>
<thead>
<tr>
<th>JOINT</th>
<th>FORCE-X/</th>
<th>FORCE-Y/</th>
<th>FORCE-Z/</th>
<th>MOM-X/</th>
<th>MOM-Y/</th>
<th>MOM-Z/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>KFX</td>
<td>KFY</td>
<td>KFZ</td>
<td>KMX</td>
<td>KMY</td>
<td>KMZ</td>
</tr>
<tr>
<td>1</td>
<td>0.0</td>
<td>85.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>0.0</td>
<td>251.5</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>0.0</td>
<td>332.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>0.0</td>
<td>332.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>0.0</td>
<td>294.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>6</td>
<td>0.0</td>
<td>265.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>
### Application Examples
EX. American Design Examples

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>138.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>170.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>502.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>587.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>531.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>276.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>170.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>502.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>587.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>531.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>276.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>22</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>165.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>488.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>24</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>646.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>25</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>646.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>26</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>570.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>27</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>515.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>28</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>268.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>29</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>160.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>30</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>473.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>31</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>626.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>32</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>626.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>33</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>553.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>34</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>500.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>35</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. American Design Examples**

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>0.0</td>
<td>260.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>37</td>
<td>0.0</td>
<td>160.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>38</td>
<td>0.0</td>
<td>473.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>39</td>
<td>0.0</td>
<td>626.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>40</td>
<td>0.0</td>
<td>626.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>41</td>
<td>0.0</td>
<td>553.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>42</td>
<td>0.0</td>
<td>500.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>43</td>
<td>0.0</td>
<td>260.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>44</td>
<td>0.0</td>
<td>140.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>45</td>
<td>0.0</td>
<td>414.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>46</td>
<td>0.0</td>
<td>548.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>47</td>
<td>0.0</td>
<td>548.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>48</td>
<td>0.0</td>
<td>484.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>49</td>
<td>0.0</td>
<td>437.5</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>50</td>
<td>0.0</td>
<td>227.5</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>51</td>
<td>0.0</td>
<td>120.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>52</td>
<td>0.0</td>
<td>355.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>53</td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>54</td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>55</td>
<td>0.0</td>
<td>415.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>56</td>
<td>0.0</td>
<td>375.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>57</td>
<td>0.0</td>
<td>195.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>58</td>
<td>0.0</td>
<td>120.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>59</td>
<td>0.0</td>
<td>355.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>60</td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>61</td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>62</td>
<td>0.0</td>
<td>415.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>63</td>
<td>0.0</td>
<td>375.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**SLAB ON GRADE**

-- PAGE NO. 4
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>SLAB ON GRADE</th>
<th>1</th>
<th>0</th>
<th>0</th>
<th>0</th>
<th>1</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.0</td>
<td>120.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>355.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>415.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>375.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>195.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>120.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>355.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>470.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>415.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>375.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>195.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>0.0</td>
<td>140.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro**

4549

User Manual
### Application Examples

#### EX. American Design Examples

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>160.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>473.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>626.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>626.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>553.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>500.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>260.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>165.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>488.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>646.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>646.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>570.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>515.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>268.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>170.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>502.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>587.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>531.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>276.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>170.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>502.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>665.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>587.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>531.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>276.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>85.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

---

**SLAB ON GRADE**

---

**PAGE NO.**  6
121 1 0 1 0 1 0
    0.0 251.5 0.0 0.0 0.0 0.0
122 1 0 1 0 1 0
    0.0 332.9 0.0 0.0 0.0 0.0
123 1 0 1 0 1 0
    0.0 332.9 0.0 0.0 0.0 0.0
124 1 0 1 0 1 0
    0.0 294.0 0.0 0.0 0.0 0.0
125 1 0 1 0 1 0
    0.0 265.6 0.0 0.0 0.0 0.0
126 1 0 1 0 1 0
    0.0 138.1 0.0 0.0 0.0 0.0

************ END OF DATA FROM INTERNAL STORAGE ************

39. LOAD 1 'WEIGHT OF MAT & EARTH'
40. ELEMENT LOAD
41. 1 TO 102 PR GY -1.55
42. LOAD 2 'COLUMN LOAD-DL+LL'
43. JOINT LOADS
44. 1 2 FY -217.
45. 8 9 FY -109.
46. 5 FY -308.7
47. 6 FY -617.4
48. 22 23 FY -410.
49. 29 30 FY -205.
50. 26 FY -542.7
51. 27 FY -1085.4
52. 43 44 50 51 71 72 78 79 FY -307.5
53. 47 54 82 FY -264.2
54. 48 55 76 83 FY -528.3
55. 92 93 FY -205.0
56. 99 100 FY -410.0
57. 103 FY -487.0
58. 104 FY -974.0
59. 113 114 FY -109.0
60. 120 121 FY -217.0
61. 124 FY -273.3
62. 125 FY -546.6
63. LOADING COMBINATION 101 TOTAL LOAD
64. 1 1. 2 1.
65. PERFORM ANALYSIS
66. LOAD LIST 101
67. PRINT JOINT DISPLACEMENTS LIST 33 56

JOINT DISPLACEMENT (INCH RADIANS) STRUCTURE TYPE = SPACE
------------------
JOINT   LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
33 101 0.00000 -4.73840 0.00000 -0.03240 0.00000 0.06201
Print Element Stresses List 34 67

Element Stresses List 34 -- Page No. 9

SLAB ON GRADE

Element Stresses

FORCE, LENGTH UNITS = KIP FEET

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT</td>
<td>VONB</td>
<td>SX</td>
<td>SY</td>
<td>SXY</td>
<td></td>
</tr>
<tr>
<td>TRESCAT</td>
<td>TRESCAB</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34 101</td>
<td>-6.17</td>
<td>-8.28</td>
<td>0.84</td>
<td>4.63</td>
<td>10.89</td>
</tr>
<tr>
<td>552.32</td>
<td>552.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>631.36</td>
<td>631.36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

TOP: SMAX = 393.78, SMIN = -237.58, TMAX = 315.68, ANGLE = 49.9
BOTT: SMAX = 72.18, SMIN = 7.72, TMAX = 15.02, ANGLE = -40.1

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT</td>
<td>VONB</td>
<td>SX</td>
<td>SY</td>
<td>SXY</td>
<td></td>
</tr>
<tr>
<td>TRESCAT</td>
<td>TRESCAB</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>67 101</td>
<td>72.18</td>
<td>7.72</td>
<td>15.02</td>
<td>9.75</td>
<td>22.08</td>
</tr>
<tr>
<td>1555.35</td>
<td>1555.35</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1269.99</td>
<td>1269.99</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

TOP: SMAX = 988.79, SMIN = -281.20, TMAX = 635.00, ANGLE = 41.6
BOTT: SMAX = 281.20, SMIN = -988.79, TMAX = 635.00, ANGLE = -48.4

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

<table>
<thead>
<tr>
<th>PLATE NO.</th>
<th>CASE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>67</td>
<td>67</td>
</tr>
<tr>
<td>67</td>
<td>101</td>
</tr>
<tr>
<td>67</td>
<td>101</td>
</tr>
<tr>
<td>67</td>
<td>101</td>
</tr>
</tbody>
</table>

END OF ELEMENT FORCES

Finish

SLAB ON GRADE

******** END OF THE STAAD.Pro RUN ********

**** DATE = APR 14, 2019 TIME = 22:56:28 ****

For technical assistance on STAAD.Pro, please visit http://www.bentley.com/en/support/
Details about additional assistance from Bentley and Partners can be found at program menu Help -> Technical Support
Copyright (c) 1997-2017 Bentley Systems, Inc.

Related Links
- M. To assign a foundation support (on page 818)
- TR.27.3 Automatic Spring Support Generator for Foundations (on page 2517)
- Create Support dialog (on page 2983)

EX. US-24 Analysis of a Concrete Block Using Solid Elements

This is an example of the analysis of a structure modeled using solid finite elements. This example also illustrates the method for applying an enforced displacement on the structure.
This problem is installed with the program by default to 
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-24 
Analysis of a Concrete Block Using Solid Elements.STD when you install the program.

STAAD SPACE 
*EXAMPLE PROBLEM USING SOLID ELEMENTS

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

UNIT KNS MET

The units for the data that follows are specified above.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0.0 0.0 2.0 4 0.0 3.0 2.0</td>
<td></td>
</tr>
<tr>
<td>5 1.0 0.0 2.0 8 1.0 3.0 2.0</td>
<td></td>
</tr>
<tr>
<td>9 2.0 0.0 2.0 12 2.0 3.0 2.0</td>
<td></td>
</tr>
<tr>
<td>21 0.0 0.0 1.0 24 0.0 3.0 1.0</td>
<td></td>
</tr>
<tr>
<td>25 1.0 0.0 1.0 28 1.0 3.0 1.0</td>
<td></td>
</tr>
<tr>
<td>29 2.0 0.0 1.0 32 2.0 3.0 1.0</td>
<td></td>
</tr>
<tr>
<td>41 0.0 0.0 0.0 44 0.0 3.0 0.0</td>
<td></td>
</tr>
<tr>
<td>45 1.0 0.0 0.0 48 1.0 3.0 0.0</td>
<td></td>
</tr>
<tr>
<td>49 2.0 0.0 0.0 52 2.0 3.0 0.0</td>
<td></td>
</tr>
</tbody>
</table>

Figure 478: Example Problem No. 24
The joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of some of those nodes are generated utilizing the fact that they are equally spaced between the extremities.

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>INCIDENCES</th>
<th>SOLID</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>5</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>9</td>
</tr>
<tr>
<td>7</td>
<td>25</td>
<td>29</td>
</tr>
</tbody>
</table>

The incidences of solid elements are defined above. The word SOLID is used to signify that these are 8-node solid elements as opposed to 3-noded or 4-noded plate elements. Each line contains the data for generating 3 elements. For example, element number 1 is first defined by all of its 8 nodes. Then, increments of 1 to the joint number and 1 to the element number (the defaults) are used for generating incidences for elements 2 and 3. Similarly, incidences of elements 4, 7 and 10 are defined while those of 5, 6, 8, 9, 11 and 12 are generated.

UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.25
DENSITY 7.5e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METERL

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

PRINT ELEMENT INFO SOLID LIST 1 TO 5

This command will enable us to obtain, in a tabular form, the details of the incidences and material property values of elements 1 to 5.

SUPPORTS
1 5 21 25 29 41 45 49 PINNED
9 ENFORCED

The above lines contain the data for supports for the model. The ENFORCED support condition is used to declare a point at which an enforced displacement load is applied later (see load case 3).

LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0

The above data describe a static load case. It consists of selfweight loading and a joint load, both in the negative global Y direction.

LOAD 2
JOINT LOADS
2 TO 4 22 TO 24 42 TO 44 FX 100.0

Load case 2 consists of several joint loads acting in the positive global X direction.

LOAD 3
SUPPORT DISPLACEMENT
9 FX 0.0011
Load case 3 consists of an enforced displacement along the global X direction at node 9. The displacement in the other enforced support degrees of freedom will default to zero.

<table>
<thead>
<tr>
<th>UNIT</th>
<th>POUND FEET</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD</td>
<td>4</td>
</tr>
<tr>
<td>ELEMENT LOAD SOLIDS</td>
<td>3 6 9 12 FACE 4 PRE GY -500.0</td>
</tr>
</tbody>
</table>

In Load case 4, a pressure load of 500 pounds/sq.ft is applied on Face # 4 of solid elements 3, 6, 9 and 12. Face 4 is defined as shown in the following table:

<table>
<thead>
<tr>
<th>Face Number</th>
<th>Surface Joints</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 front</td>
<td>Jt 1, Jt 4, Jt 3, Jt 2</td>
</tr>
<tr>
<td>2 bottom</td>
<td>Jt 1, Jt 2, Jt 6, Jt 5</td>
</tr>
<tr>
<td>3 left</td>
<td>Jt 1, Jt 5, Jt 8, Jt 4</td>
</tr>
<tr>
<td>4 top</td>
<td>Jt 4, Jt 8, Jt 7, Jt 3</td>
</tr>
<tr>
<td>5 right</td>
<td>Jt 2, Jt 3, Jt 7, Jt 6</td>
</tr>
<tr>
<td>6 back</td>
<td>Jt 5, Jt 6, Jt 7, Jt 8</td>
</tr>
</tbody>
</table>

The above table, and other details of this type of loading can be found in section 5.32.3.2 of the STAAD.Pro Technical Reference manual (on page 2660).

<table>
<thead>
<tr>
<th>UNIT</th>
<th>KNS MMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD</td>
<td>5</td>
</tr>
<tr>
<td>REPEAT LOAD</td>
<td>1 1.0 2 1.0 3 1.0 4 1.0</td>
</tr>
</tbody>
</table>

Load case 5 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We want the program to analyze the structure for loads from cases 1 through 4 acting simultaneously. In other words, the above instruction is the same as the following:

```
LOAD 5
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
2 TO 4 22 TO 24 42 TO 44 FX 100.0
SUPPORT DISPLACEMENT
9 FX .0011
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
LOAD COMB 10
1 1.0 2 1.0
```

Load case 10 is a combination load case, which combines the effects of cases 1 & 2. While the syntax of this might look very similar to that of the REPEAT LOAD case shown in case 5, there is a fundamental difference. In a REPEAT LOAD case, the program computes the displacements by multiplying the inverted stiffness matrix by the
load vector built for the REPEAT LOAD case. But in solving load combination cases, the program merely calculates the end results (displacements, forces, reactions) by gathering up the corresponding values from the individual components of the combination case, factoring them, and then algebraically summing them up. This difference in approach is quite important in that non-linear problems such as PDELTA ANALYSIS, MEMBER TENSION, and MEMBER COMPRESSION situations, changes in support conditions etc. should be handled using REPEAT LOAD cases, not load combination cases.

PERFORM ANALYSIS PRINT STATICS CHECK

A static equilibrium report, consisting of total applied loading and total support reactions from each primary load case is requested along with the instructions to carry out a linear static analysis.

PRINT JOINT DISPLACEMENTS LIST 8 9

Global displacements at nodes 8 and 9 are obtained using the above command.

UNIT KNS METER
PRINT SUPPORT REACTIONS

Reactions at the supports are obtained using the above command.

UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6

This command requests the program to provide the element stress results at the nodes of elements 4 and 6. The results will be printed for all the load cases. The word SOLID is used to signify that these are solid elements as opposed to plate or shell elements.

FINISH

The STAAD run is terminated.

Input File

STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
UNIT KNS MET
JOINT COORDINATES

GLOBAL DISPLACEMENTS

UNIT KNS METER
PRINT SUPPORT REACTIONS

Reactions at the supports are obtained using the above command.

UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6

This command requests the program to provide the element stress results at the nodes of elements 4 and 6. The results will be printed for all the load cases. The word SOLID is used to signify that these are solid elements as opposed to plate or shell elements.

FINISH

The STAAD run is terminated.

Input File

STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
UNIT KNS MET
JOINT COORDINATES

Global displacements at nodes 8 and 9 are obtained using the above command.

UNIT KNS METER
PRINT SUPPORT REACTIONS

Reactions at the supports are obtained using the above command.

UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6

This command requests the program to provide the element stress results at the nodes of elements 4 and 6. The results will be printed for all the load cases. The word SOLID is used to signify that these are solid elements as opposed to plate or shell elements.

FINISH

The STAAD run is terminated.
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
PRINT ELEMENT INFO SOLID LIST 1 TO 5
SUPPORTS
1 5 21 25 29 41 45 49 PINNED
9 ENFORCED BUT MX MY MZ
LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
LOAD 2
JOINT LOADS
2 TO 4 22 TO 24 42 TO 44 FX 100.0
LOAD 3
SUPPORT DISPLACEMENT
9 FX .0011
UNIT POUND FEET
LOAD 4
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
UNIT KNS MMS
LOAD 5
REPEAT LOAD
1.0 2.0 1.0 3.0 1.0 4.0 1.0
LOAD COMB 10
1.0 2.1.0
PERFORM ANALYSIS PRINT STAT CHECK
PRINT JOINT DISPLACEMENTS LIST 8 9
UNIT KNS METER
PRINT SUPPORT REACTIONS
UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
FINISH

STAAD Output File

1. STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
INPUT FILE: US-24 Analysis of a Concrete Block Using Solid Elements.STD
2. UNIT KNS MET
3. JOINT COORDINATES
4. 1 0.0 0.0 2.0 4 0.0 3.0 2.0
5. 5 1.0 0.0 2.0 8 1.0 3.0 2.0
6. 9 2.0 0.0 2.0 12 2.0 3.0 2.0

Application Examples
EX. American Design Examples
### Example Problem Using Solid Elements

#### Example Problem Using Solid Elements -- Page No. 2

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>NODE-1</th>
<th>NODE-2</th>
<th>NODE-3</th>
<th>NODE-4</th>
<th>NODE-5</th>
<th>NODE-6</th>
<th>NODE-7</th>
<th>NODE-8</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>5</td>
<td>6</td>
<td>2</td>
<td>21</td>
<td>25</td>
<td>26</td>
<td>22</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>6</td>
<td>7</td>
<td>3</td>
<td>22</td>
<td>26</td>
<td>27</td>
<td>23</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>7</td>
<td>8</td>
<td>4</td>
<td>23</td>
<td>27</td>
<td>28</td>
<td>24</td>
</tr>
<tr>
<td>4</td>
<td>21</td>
<td>25</td>
<td>26</td>
<td>22</td>
<td>41</td>
<td>45</td>
<td>46</td>
<td>42</td>
</tr>
<tr>
<td>5</td>
<td>22</td>
<td>26</td>
<td>27</td>
<td>23</td>
<td>42</td>
<td>46</td>
<td>47</td>
<td>43</td>
</tr>
</tbody>
</table>

#### Example Problem Using Solid Elements -- Page No. 3

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>YOUNG'S MODULUS</th>
<th>MODULUS OF RIGIDITY</th>
<th>DENSITY</th>
<th>ALPHA</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.1000002E+08</td>
<td>0.0000000E+00</td>
<td>7.5000E+01</td>
<td>6.0000E-06</td>
</tr>
<tr>
<td>2</td>
<td>2.1000002E+08</td>
<td>0.0000000E+00</td>
<td>7.5000E+01</td>
<td>6.0000E-06</td>
</tr>
<tr>
<td>3</td>
<td>2.1000002E+08</td>
<td>0.0000000E+00</td>
<td>7.5000E+01</td>
<td>6.0000E-06</td>
</tr>
<tr>
<td>4</td>
<td>2.1000002E+08</td>
<td>0.0000000E+00</td>
<td>7.5000E+01</td>
<td>6.0000E-06</td>
</tr>
<tr>
<td>5</td>
<td>2.1000002E+08</td>
<td>0.0000000E+00</td>
<td>7.5000E+01</td>
<td>6.0000E-06</td>
</tr>
</tbody>
</table>

#### Supports

<table>
<thead>
<tr>
<th>SUPPORTS</th>
<th>NODES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5</td>
</tr>
<tr>
<td>2</td>
<td>21</td>
</tr>
<tr>
<td>3</td>
<td>25</td>
</tr>
<tr>
<td>4</td>
<td>29</td>
</tr>
<tr>
<td>5</td>
<td>41</td>
</tr>
<tr>
<td>6</td>
<td>45</td>
</tr>
<tr>
<td>7</td>
<td>49</td>
</tr>
</tbody>
</table>

#### Load 1

<table>
<thead>
<tr>
<th>SELF Y</th>
<th>JOINT LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1.0</td>
<td></td>
</tr>
</tbody>
</table>

#### Load 2

<table>
<thead>
<tr>
<th>JOINT LOADS</th>
<th>LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 TO 4</td>
<td>42 TO 44</td>
</tr>
<tr>
<td>FX 100.0</td>
<td></td>
</tr>
</tbody>
</table>

#### Load 3

<table>
<thead>
<tr>
<th>SUPPORT DISPLACEMENT</th>
<th>LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 FX 0.011</td>
<td></td>
</tr>
</tbody>
</table>

#### Unit Pounds Feet

---

*Some Additional Features*

### Constants

#### Material Steel All

<table>
<thead>
<tr>
<th>UNIT</th>
<th>PRINT ELEMENT INFO SOLID LIST 1 TO 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Meter</td>
<td>E 210</td>
</tr>
<tr>
<td></td>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td></td>
<td>E 210</td>
</tr>
<tr>
<td></td>
<td>POISSON 0.25</td>
</tr>
<tr>
<td></td>
<td>DENSITY 7.5E-008</td>
</tr>
<tr>
<td></td>
<td>ALPHA 6E-006</td>
</tr>
<tr>
<td></td>
<td>DAMP 0.03</td>
</tr>
<tr>
<td></td>
<td>TYPE STEEL</td>
</tr>
<tr>
<td></td>
<td>STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2</td>
</tr>
</tbody>
</table>

---

*Example Problem Using SOLID ELEMENTS*
47. LOAD 4
48. ELEMENT LOAD SOLIDS
49. 3 6 9 12 FACE 4 PRE GY -500.0
50. UNIT KNS MMS
51. LOAD 5
52. REPEAT LOAD
53. 1 1.0 2 1.0 3 1.0 4 1.0
54. LOAD COMB 10
55. 1 1.0 2 1.0
56. PERFORM ANALYSIS PRINT STAT CHECK

PROBLEM STATISTICS
----------------------------
NUMBER OF JOINTS         36  NUMBER OF MEMBERS       0
NUMBER OF PLATES          0  NUMBER OF SOLIDS       12
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      9

EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO. 4
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     5, TOTAL DEGREES OF FREEDOM =      84
TOTAL LOAD COMBINATION CASES =     1  SO FAR.

EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO. 5
STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 1
CENTER OF FORCE BASED ON Y FORCES ONLY (MMS).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.999999993E+03
Y =  0.228947364E+04
Z =  0.999999993E+03

TOTAL APPLIED LOAD  1
***TOTAL APPLIED LOAD ( KNS  MMS ) SUMMARY (LOADING  1 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =       -1900.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=     1900000.15  MY=           0.00  MZ=    -1900000.15

TOTAL REACTION LOAD  1
***TOTAL REACTION LOAD( KNS  MMS ) SUMMARY (LOADING  1 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =        1900.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=    -1900000.15  MY=          -0.00  MZ=     1900000.15

MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING  1)
MAXIMUMS AT NODE
X = -1.21106E-04     23
Y = -1.15439E-03     28
Z =  1.21106E-04     7
RX=  0.00000E+00     0
RY=  0.00000E+00     0
RZ=  0.00000E+00     0

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.  2
CENTER OF FORCE BASED ON X FORCES ONLY (MMS).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.000000000E+00
Y =  0.199999999E+04
Z =  0.999999993E+03

EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO. 6
TOTAL APPLIED LOAD  2
***TOTAL APPLIED LOAD ( KNS  MMS ) SUMMARY (LOADING  2 )
### Application Examples

**EX. American Design Examples**

**STAAD.Pro 4560 User Manual**

<table>
<thead>
<tr>
<th>Case No.</th>
<th>Total Applied Load</th>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>SUMMATION FORCE-X = 900.00</td>
<td>MX= 0.00 MY= 900000.03 MZ= -1800000.06</td>
</tr>
<tr>
<td>2</td>
<td>SUMMATION FORCE-Y = 0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>SUMMATION FORCE-Z = 0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>SUMMATION OF MOMENTS AROUND THE ORIGIN- MX= 0.00 MY= 900000.03 MZ= 1800000.06</td>
<td></td>
</tr>
</tbody>
</table>

**Total Reaction Load 2**

<table>
<thead>
<tr>
<th>Maximum Displacements (CM /RADIANS) (LOADING 2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>X = 2.22892E-03 4</td>
</tr>
<tr>
<td>Y = 7.83934E-04 44</td>
</tr>
<tr>
<td>Z = 9.49033E-05 10</td>
</tr>
<tr>
<td>RX= 0.00000E+00 0</td>
</tr>
<tr>
<td>RY= 0.00000E+00 0</td>
</tr>
<tr>
<td>RZ= 0.00000E+00 0</td>
</tr>
</tbody>
</table>

**Static Load/Reaction/Equilibrium Summary for Case No. 3**

<table>
<thead>
<tr>
<th>Total Applied Load</th>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>SUMMATION FORCE-X = 0.0000000E+00</td>
</tr>
<tr>
<td>3</td>
<td>SUMMATION FORCE-Y = 0.0000000E+00</td>
</tr>
<tr>
<td>3</td>
<td>SUMMATION FORCE-Z = 0.0000000E+00</td>
</tr>
<tr>
<td>3</td>
<td>SUMMATION OF MOMENTS AROUND THE ORIGIN- MX= 0.0000000E+00 MY= 0.0000000E+00 MZ= 0.0000000E+00</td>
</tr>
</tbody>
</table>

**Total Reaction Load 3**

<table>
<thead>
<tr>
<th>Maximum Displacements (CM /RADIANS) (LOADING 3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>X = 1.10000E-01 9</td>
</tr>
<tr>
<td>Y = -1.21497E-02 6</td>
</tr>
<tr>
<td>Z = 1.61372E-02 24</td>
</tr>
<tr>
<td>RX= 0.00000E+00 0</td>
</tr>
<tr>
<td>RY= 0.00000E+00 0</td>
</tr>
<tr>
<td>RZ= 0.00000E+00 0</td>
</tr>
</tbody>
</table>

**Example Problem Using Solid Elements**

**Center of Force Based on Y Forces Only (MMS).**

** Forces in Non-Global Directions Will Invalidate Results**

<table>
<thead>
<tr>
<th>Total Applied Load</th>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>SUMMATION FORCE-X = 0.00</td>
</tr>
<tr>
<td>4</td>
<td>SUMMATION FORCE-Y = -95.76</td>
</tr>
<tr>
<td>4</td>
<td>SUMMATION FORCE-Z = 0.00</td>
</tr>
<tr>
<td>4</td>
<td>SUMMATION OF MOMENTS AROUND THE ORIGIN- MX= 95760.52 MY= 0.00 MZ= -95760.52</td>
</tr>
</tbody>
</table>

**Total Reaction Load 4**
**Application Examples**

**EX. American Design Examples**

---

**TOTAL REACTION LOAD (KNS MMS) SUMMARY (LOADING 4)**

- **SUMMATION FORCE-X**: 0.00
- **SUMMATION FORCE-Y**: 95.76
- **SUMMATION FORCE-Z**: 0.00

**SUMMATION OF MOMENTS AROUND THE ORIGIN**

- **MX**: -95760.52
- **MY**: 0.00
- **MZ**: 95760.52

**MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 4)**

- **MAXIMUMS AT NODE**
  - **X**: 3.17652E-06
  - **Y**: -3.35288E-05
  - **Z**: -3.17652E-06
  - **RX**: 0.00000E+00
  - **RY**: 0.00000E+00
  - **RZ**: 0.00000E+00

**TOTAL APPLIED LOAD (KNS MMS) SUMMARY (LOADING 5)**

- **SUMMATION FORCE-X**: 900.00
- **SUMMATION FORCE-Y**: -1995.76
- **SUMMATION FORCE-Z**: 0.00

**SUMMATION OF MOMENTS AROUND THE ORIGIN**

- **MX**: 1995760.67
- **MY**: 900000.03
- **MZ**: -3795760.73

**TOTAL REACTION LOAD (KNS MMS) SUMMARY (LOADING 5)**

- **SUMMATION FORCE-X**: -900.00
- **SUMMATION FORCE-Y**: 1995.76
- **SUMMATION FORCE-Z**: 0.00

**SUMMATION OF MOMENTS AROUND THE ORIGIN**

- **MX**: -1995760.67
- **MY**: -900000.03
- **MZ**: 3795760.73

**MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 5)**

- **MAXIMUMS AT NODE**
  - **X**: 1.10000E-01
  - **Y**: -1.23568E-02
  - **Z**: 1.61372E-02
  - **RX**: 0.00000E+00
  - **RY**: 0.00000E+00
  - **RZ**: 0.00000E+00

---

**Example Problem Using Solid Elements -- Page No. 8**

**CENTER OF FORCE BASED ON X FORCES ONLY (MMS).**

(Forces in non-global directions will invalidate results)

- **X**: 0.000000000E+00
- **Y**: 0.199999999E+04
- **Z**: 0.999999993E+03

**CENTER OF FORCE BASED ON Y FORCES ONLY (MMS).**

(Forces in non-global directions will invalidate results)

- **X**: 0.999999993E+03
- **Y**: 0.232356609E+04
- **Z**: 0.999999993E+03

**TOTAL APPLIED LOAD 5**

**TOTAL REACTION LOAD 5**

---

**JOINT DISPLACEMENT LIST 8 9**

**JOINT DISPLACEMENT (CM/RADIANS) STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>1</td>
<td>0.0000</td>
<td>-0.0002</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>0.0020</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0.0193</td>
<td>-0.0049</td>
<td>0.0089</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

---

*STAAD.Pro 4561 User Manual*
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>27.47</td>
<td>128.97</td>
<td>-27.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-72.24</td>
<td>-232.67</td>
<td>42.18</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-2022.70</td>
<td>-302.04</td>
<td>-1192.39</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1.52</td>
<td>6.63</td>
<td>-1.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-2065.94</td>
<td>-399.11</td>
<td>-1179.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-44.76</td>
<td>-103.70</td>
<td>14.70</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### Example Problem Using Solid Elements

---

**SUPPORT REACTIONS**

**Example Problem Using Solid Elements**

**Structure Type = SPACE**

---

**JOINT LOAD**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>27.47</td>
<td>128.97</td>
<td>-27.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-72.24</td>
<td>-232.67</td>
<td>42.18</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-891.15</td>
<td>-2739.86</td>
<td>-1598.54</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1.52</td>
<td>6.63</td>
<td>-1.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-934.39</td>
<td>-2836.93</td>
<td>-1611.72</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-44.76</td>
<td>-103.70</td>
<td>14.70</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**Application Examples**

**EX. American Design Examples**

---

**STAAD.Pro User Manual**
### Example Problem Using Solid Elements

**Support Reactions**

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>-888.61</td>
<td>2435.62</td>
<td>1266.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>-27.47</td>
<td>128.97</td>
<td>-27.47</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-81.35</td>
<td>226.24</td>
<td>-45.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>39169.49</td>
<td>-8551.31</td>
<td>13133.66</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-1.52</td>
<td>6.63</td>
<td>-1.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Example Problem Using Solid Elements**

**Joint Load Center**

<table>
<thead>
<tr>
<th>NODE/ ELEMENT LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 1 21</td>
<td>-0.144</td>
<td>-0.449</td>
<td>-0.155</td>
<td>-0.006</td>
<td>-0.011</td>
<td>0.000</td>
</tr>
<tr>
<td>4 1 25</td>
<td>-0.132</td>
<td>-0.368</td>
<td>-0.132</td>
<td>-0.011</td>
<td>-0.011</td>
<td>0.005</td>
</tr>
<tr>
<td>4 1 26</td>
<td>-0.009</td>
<td>-0.377</td>
<td>-0.009</td>
<td>-0.003</td>
<td>-0.003</td>
<td>0.005</td>
</tr>
<tr>
<td>4 1 22</td>
<td>-0.012</td>
<td>-0.449</td>
<td>-0.005</td>
<td>0.002</td>
<td>-0.018</td>
<td>0.009</td>
</tr>
<tr>
<td>4 1 41</td>
<td>-0.152</td>
<td>-0.484</td>
<td>-0.152</td>
<td>-0.015</td>
<td>-0.015</td>
<td>-0.005</td>
</tr>
<tr>
<td>4 1 45</td>
<td>-0.155</td>
<td>-0.449</td>
<td>-0.144</td>
<td>-0.011</td>
<td>-0.006</td>
<td>-0.000</td>
</tr>
<tr>
<td>4 1 46</td>
<td>-0.005</td>
<td>-0.449</td>
<td>-0.012</td>
<td>-0.018</td>
<td>-0.002</td>
<td>0.009</td>
</tr>
<tr>
<td>4 1 42</td>
<td>0.007</td>
<td>-0.475</td>
<td>0.007</td>
<td>-0.023</td>
<td>-0.023</td>
<td>0.014</td>
</tr>
<tr>
<td>4 1 CENTER</td>
<td>-0.075</td>
<td>-0.437</td>
<td>-0.075</td>
<td>-0.011</td>
<td>-0.011</td>
<td>0.005</td>
</tr>
</tbody>
</table>

**Normal Stresses**

- **S1= -0.070**
- **S2= -0.080**
- **S3= -0.438**
- **SE= 0.363**

**Application Examples**

**Example Problem Using Solid Elements**

**Joint Load Center**

<table>
<thead>
<tr>
<th>NODE/ ELEMENT LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 3 21</td>
<td>0.900</td>
<td>5.181</td>
<td>1.884</td>
<td>4.989</td>
<td>5.058</td>
<td>0.396</td>
</tr>
<tr>
<td>4 3 25</td>
<td>-0.893</td>
<td>-5.740</td>
<td>-1.294</td>
<td>6.615</td>
<td>1.429</td>
<td>-1.229</td>
</tr>
<tr>
<td>4 3 26</td>
<td>5.251</td>
<td>-3.282</td>
<td>3.647</td>
<td>5.654</td>
<td>0.468</td>
<td>3.274</td>
</tr>
</tbody>
</table>
### EXAMPLE PROBLEM USING SOLID ELEMENTS

**ELEMENT STRESSES**
**UNITS= NEWTMMS**

<table>
<thead>
<tr>
<th>NODE/ ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SXX</td>
<td>SYX</td>
</tr>
<tr>
<td></td>
<td>SYY</td>
<td>SYZ</td>
</tr>
<tr>
<td></td>
<td>SZZ</td>
<td>SZX</td>
</tr>
</tbody>
</table>

| 4 3 CENTER | 1.325 | 3.324 | 1.517 | 2.548 | 3.244 | 1.023 |

**S1= 7.030**  **S2= 0.411**  **S3= -1.275**  **SE= 7.604**
**DC= 0.425**  **0.744**  **0.516**  **0.089**  **-0.055**  **-0.586**

<table>
<thead>
<tr>
<th>4 3 22</th>
<th>5.379</th>
<th>5.974</th>
<th>1.830</th>
<th>4.029</th>
<th>6.019</th>
<th>1.649</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 3 41</td>
<td>2.148</td>
<td>9.507</td>
<td>2.550</td>
<td>0.107</td>
<td>5.891</td>
<td>1.229</td>
</tr>
<tr>
<td>4 3 45</td>
<td>2.276</td>
<td>4.348</td>
<td>1.292</td>
<td>-1.518</td>
<td>0.596</td>
<td>-0.364</td>
</tr>
<tr>
<td>4 3 46</td>
<td>-1.334</td>
<td>3.555</td>
<td>2.982</td>
<td>-0.558</td>
<td>-0.364</td>
<td></td>
</tr>
<tr>
<td>4 3 42</td>
<td>-3.127</td>
<td>7.049</td>
<td>-0.756</td>
<td>1.067</td>
<td>6.851</td>
<td>0.816</td>
</tr>
</tbody>
</table>

| 4 3 CENTER | 1.325 | 3.324 | 1.517 | 2.548 | 3.244 | 1.023 |

**S1= 7.030**  **S2= 0.411**  **S3= -1.275**  **SE= 7.604**
**DC= 0.425**  **0.744**  **0.516**  **0.089**  **-0.055**  **-0.586**

<table>
<thead>
<tr>
<th>4 4 21</th>
<th>-0.008</th>
<th>-0.024</th>
<th>-0.008</th>
<th>-0.001</th>
<th>-0.001</th>
<th>-0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 4 25</td>
<td>-0.008</td>
<td>-0.022</td>
<td>-0.008</td>
<td>-0.001</td>
<td>-0.001</td>
<td>-0.000</td>
</tr>
<tr>
<td>4 4 26</td>
<td>0.001</td>
<td>-0.022</td>
<td>0.001</td>
<td>-0.001</td>
<td>-0.001</td>
<td>-0.000</td>
</tr>
<tr>
<td>4 4 41</td>
<td>-0.008</td>
<td>-0.026</td>
<td>-0.008</td>
<td>-0.001</td>
<td>-0.001</td>
<td>-0.000</td>
</tr>
<tr>
<td>4 4 45</td>
<td>-0.008</td>
<td>-0.024</td>
<td>-0.008</td>
<td>-0.001</td>
<td>-0.001</td>
<td>-0.000</td>
</tr>
<tr>
<td>4 4 46</td>
<td>0.001</td>
<td>-0.024</td>
<td>0.001</td>
<td>-0.001</td>
<td>-0.001</td>
<td>-0.000</td>
</tr>
<tr>
<td>4 4 CENTER</td>
<td>-0.004</td>
<td>-0.024</td>
<td>-0.004</td>
<td>-0.001</td>
<td>-0.001</td>
<td>-0.000</td>
</tr>
</tbody>
</table>

**S1= -0.003**  **S2= -0.004**  **S3= -0.024**  **SE= 0.021**
**DC= 0.705**  **-0.070**  **0.705**  **-0.707**  **0.000**  **0.707**

<table>
<thead>
<tr>
<th>4 5 21</th>
<th>0.925</th>
<th>5.729</th>
<th>2.005</th>
<th>5.199</th>
<th>5.061</th>
<th>0.402</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 5 25</td>
<td>-0.878</td>
<td>-6.136</td>
<td>-1.412</td>
<td>6.854</td>
<td>1.431</td>
<td>-1.254</td>
</tr>
<tr>
<td>4 5 26</td>
<td>5.215</td>
<td>-3.629</td>
<td>3.624</td>
<td>5.903</td>
<td>0.481</td>
<td>3.277</td>
</tr>
<tr>
<td>4 5 22</td>
<td>5.313</td>
<td>6.532</td>
<td>1.928</td>
<td>4.248</td>
<td>6.011</td>
<td>1.622</td>
</tr>
</tbody>
</table>

**S1= 7.193**  **S2= 0.379**  **S3= -1.400**  **SE= 7.856**

**EX. American Design Examples**
### Application Examples

**EX. American Design Examples**

#### EXAMPLE PROBLEM USING SOLID ELEMENTS

**ELEMENT LOAD CENTER**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>1</td>
<td>44</td>
<td>-0.259</td>
<td>-0.089</td>
<td>-0.259</td>
<td>0.273</td>
<td>0.273</td>
</tr>
<tr>
<td>S1=</td>
<td>0.027</td>
<td>S2=</td>
<td>-0.044</td>
<td>S3=</td>
<td>-0.368</td>
<td>SE=</td>
<td>0.365</td>
</tr>
<tr>
<td>DC=</td>
<td>0.631</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>23</td>
<td>-0.032</td>
<td>0.112</td>
<td>-0.001</td>
<td>0.030</td>
<td>0.002</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>27</td>
<td>-0.001</td>
<td>0.025</td>
<td>-0.046</td>
<td>0.073</td>
<td>0.013</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>28</td>
<td>-0.096</td>
<td>-0.003</td>
<td>-0.065</td>
<td>0.083</td>
<td>0.003</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>24</td>
<td>-0.085</td>
<td>0.177</td>
<td>0.109</td>
<td>0.040</td>
<td>0.012</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>43</td>
<td>-0.152</td>
<td>0.158</td>
<td>0.052</td>
<td>0.136</td>
<td>0.023</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>47</td>
<td>-0.140</td>
<td>-0.041</td>
<td>-0.013</td>
<td>0.092</td>
<td>0.008</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>48</td>
<td>-0.496</td>
<td>-0.105</td>
<td>-0.119</td>
<td>0.082</td>
<td>0.019</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>CENTER</td>
<td>-0.183</td>
<td>0.051</td>
<td>-0.001</td>
<td>0.083</td>
<td>-0.007</td>
</tr>
<tr>
<td>S1=</td>
<td>0.081</td>
<td>S2=</td>
<td>-0.001</td>
<td>S3=</td>
<td>-0.213</td>
<td>SE=</td>
<td>0.263</td>
</tr>
<tr>
<td>DC=</td>
<td>0.314</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>23</td>
<td>-2.744</td>
<td>-0.535</td>
<td>-0.041</td>
<td>-0.327</td>
<td>-0.468</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>27</td>
<td>-3.140</td>
<td>-0.556</td>
<td>-1.018</td>
<td>0.642</td>
<td>0.296</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>28</td>
<td>1.815</td>
<td>0.568</td>
<td>0.607</td>
<td>0.402</td>
<td>0.056</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>24</td>
<td>1.900</td>
<td>0.279</td>
<td>0.654</td>
<td>-0.567</td>
<td>-0.228</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>43</td>
<td>0.636</td>
<td>-0.478</td>
<td>0.687</td>
<td>0.031</td>
<td>0.313</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>47</td>
<td>0.721</td>
<td>0.942</td>
<td>0.191</td>
<td>0.999</td>
<td>0.141</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>48</td>
<td>-0.136</td>
<td>0.128</td>
<td>-0.121</td>
<td>0.759</td>
<td>0.099</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>44</td>
<td>-0.531</td>
<td>-1.602</td>
<td>-0.555</td>
<td>0.218</td>
<td>0.073</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>CENTER</td>
<td>-0.185</td>
<td>0.157</td>
<td>0.085</td>
<td>0.179</td>
<td>0.086</td>
</tr>
<tr>
<td>S1=</td>
<td>0.143</td>
<td>S2=</td>
<td>-0.010</td>
<td>S3=</td>
<td>-0.424</td>
<td>SE=</td>
<td>0.508</td>
</tr>
<tr>
<td>DC=</td>
<td>-0.484</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>23</td>
<td>0.296</td>
<td>0.507</td>
<td>0.412</td>
<td>-0.013</td>
<td>-0.128</td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>27</td>
<td>-0.071</td>
<td>-1.764</td>
<td>-0.117</td>
<td>0.025</td>
<td>0.112</td>
</tr>
</tbody>
</table>

**ELEMENT STRESSES**

**UNITS= NEWTMMS**

**NODE/ ELEMENT LOAD CENTER**

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>4</td>
<td>23</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>27</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>28</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>24</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>43</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>47</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>48</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>CENTER</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>S1=</td>
<td>0.000</td>
<td>S2=</td>
<td>-0.000</td>
<td>S3=</td>
<td>-0.024</td>
<td>SE=</td>
<td>0.024</td>
</tr>
<tr>
<td>DC=</td>
<td>-0.707</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>10</td>
<td>23</td>
<td>0.296</td>
<td>0.507</td>
<td>0.412</td>
<td>-0.013</td>
<td>-0.128</td>
</tr>
<tr>
<td>6</td>
<td>10</td>
<td>27</td>
<td>0.071</td>
<td>-1.764</td>
<td>-0.117</td>
<td>0.025</td>
<td>0.112</td>
</tr>
</tbody>
</table>

STAAD.Pro

4565

User Manual
Related Links

- **M. To assign an enforced support** (on page 813)
- **TR.27.1 Global Support Specification** (on page 2514)
- **Create Support dialog** (on page 2983)

**EX. US-25 Analysis of a Structure with Compression-Only Members**

This example demonstrates the usage of compression-only members. Since the structural condition is load dependent, the PERFORM ANALYSIS command is specified once for each primary load case.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-25 Analysis of a Structure with Compression-Only Members.STD when you install the program.
This example has been created to illustrate the command specification for a structure with certain members capable of carrying compressive force only. It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of “active” members (and hence the stiffness matrix) is load case dependent.

where:

\[
\begin{align*}
L &= 15 \text{ ft, } H = 10 \text{ ft} \\
\text{Load case 1: } P_1 &= 10 \text{ kips} & P_2 &= 15 \text{ kips} \\
\text{Load case 2: } P_3 &= -10 \text{ kips} & P_4 &= -15 \text{ kips}
\end{align*}
\]

**STAAD PLANE**
* EXAMPLE FOR COMPRESSION-ONLY MEMBERS

The input data is initiated with the word STAAD. This structure is a PLANE frame. The second line is an optional comment line.

**UNIT FEET KIP**

Units for the commands to follow are specified above.

**JOINT COORDINATES**

1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0

* Figure 479: Example Problem No. 25

**Application Examples**
EX. American Design Examples
Joint coordinates of joints 1 to 6 are defined above.

MEMBER INCIDENCES
1 1 2 5 ; 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5

The members 1 to 10 are defined along with the joints to which they are connected.

MEMBER COMPRESSION
6 TO 9

Members 6 to 9 are defined as COMPRESSION-only members. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a tensile force, it is disabled from the structure and the analysis is carried out again with the modified structure.

MEMBER PROPERTY AMERICAN
1 TO 10 TA ST W12X26

Properties for members 1 to 10 are defined as the STandard W12X26 section from the American AISC steel table.

MEMBER PROPERTY BRITISH
1 TO 10 TA ST UC152X152X30

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

SUPPORT
1 6 PINNED

Joints 1 and 6 are declared as pinned-supported.

LOAD 1
JOINT LOAD
2 FX 15
3 FX 10

Load 1 is defined above and consists of joint loads in the global X direction at joints 2 and 3.

PERFORM ANALYSIS

The above structure is analyzed for load case 1.

CHANGE
MEMBER COMPRESSION
6 TO 9

One or more among the members 6 to 9 may have been in-activated in the previous analysis. The CHANGE command restores the original structure to prepare it for the analysis for the next primary load case. The members with the compression-only attribute are specified again.

LOAD 2
JOINT LOAD
In load case 2, joint loads are applied in the negative global X direction at joints 4 and 5.

The instruction to analyze the structure is specified again. Next, any compression-only members that were inactivated during the second analysis (due to the fact that they were subjected to tensile axial forces) are re-activated with the CHANGE command. Without the re-activation, these members cannot be accessed for further processing.

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously. In other words, the above instruction is the same as the following:

The analysis is carried out for load case 3.

The members inactivated during the analysis of load case 3 are re-activated for further processing.

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the “active” load cases. The LOAD LIST ALL command enables all the load cases in the structure to be made active for further processing.

The program is instructed to write the joint displacements, support reactions and member forces to the output file.

The STAAD run is terminated.

Input File

```
STAAD PLANE EXAMPLE FOR COMPRESSION-ONLY MEMBERS
UNIT FEET KIP
SET NL 3
JOINT COORDINATES
1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
MEMBER COMPRESSION
```

6 TO 9
MEMBER PROPERTY AMERICAN
1 TO 10 TA ST W12X26
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 6 PINNED
LOAD 1
JOINT LOAD
2 FX 15
3 FX 10
PERFORM ANALYSIS
CHANGE
MEMBER COMPRESSION
6 TO 9
LOAD 2
JOINT LOAD
4 FX -10
5 FX -15
PERFORM ANALYSIS
CHANGE
MEMBER COMPRESSION
6 TO 9
LOAD 3
REPEAT LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ANALYSIS RESULTS
FINISH

STAAD Output File

******************************************************************************
* STAAD.Pro CONNECT Edition                                          *
* Version 22.01.00.*                                               *
* Proprietary Program of                                           *
* Bentley Systems, Inc.                                          *
* Date= APR 14, 2019                                         *
* Time= 22:56:37                                               *
* Licensed to: Bentley Systems Inc                                *
******************************************************************************
1. STAAD PLANE EXAMPLE FOR COMPRESSION-ONLY MEMBERS
INPUT FILE: US-25 Analysis of a Structure with Compression-Only Members.STD
2. UNIT FEET KIP
3. SET NL 3
4. JOINT COORDINATES
5. 1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
6. MEMBER INCIDENCES
7. 1 1 2 5
8. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
9. MEMBER COMPRESSION
10. 6 TO 9
11. MEMBER PROPERTY AMERICAN
12. 1 TO 10 TA ST W12X26
13. UNIT INCH
14. DEFINE MATERIAL START
15. ISOTROPIC STEEL
16. E 29000
17. POISSON 0.3
18. DENSITY 283E-006
19. ALPHA 6E-006
20. DAMP 0.03
21. TYPE STEEL
22. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
23. END DEFINE MATERIAL
24. CONSTANT
25. MATERIAL STEEL ALL
26. SUPPORT
27. 1 6 PINNED
28. LOAD 1
29. JOINT LOAD
30. 2 FX 15
31. 3 FX 10
32. PERFORM ANALYSIS

EXAMPLE FOR COMPRESSION-ONLY MEMBERS -- PAGE NO. 2

PROBLEM STATISTICS

-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS   10
NUMBER OF PLATES          0  NUMBER OF SOLIDS     0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS  2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
*** LOAD CASE     1 -- START ITERATION NO.   2
**NOTE-Tension/Compression converged after  2 iterations, Case=    1

33. CHANGE
34. MEMBER COMPRESSION
35. 6 TO 9
36. LOAD 2
37. JOINT LOAD
38. 4 FX -10
39. 5 FX -15
40. PERFORM ANALYSIS

*** LOAD CASE     2 -- START ITERATION NO.   2
**NOTE-Tension/Compression converged after  2 iterations, Case=    2

41. CHANGE
42. MEMBER COMPRESSION
43. 6 TO 9
44. LOAD 3
Application Examples
EX. American Design Examples

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0005</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.0431</td>
<td>0.0126</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.0509</td>
<td>0.0037</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.0775</td>
<td>0.0162</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.0776</td>
<td>-0.0038</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.0025</td>
<td>0.0215</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| SUPPORT REACTIONS -UNIT KIP INCH STRUCTURE TYPE = PLANE
<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-0.13</td>
<td>-23.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>24.87</td>
<td>23.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>6.90</td>
<td>-0.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>15.87</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-6.90</td>
<td>0.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.90</td>
<td>0.00</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

| MEMBER END FORCES STRUCTURE TYPE = PLANE
<table>
<thead>
<tr>
<th>MEMBER LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-23.33</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>23.33</td>
<td>-0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>15.87</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>6.90</td>
<td>-0.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.90</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-1.47</td>
<td>0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>3.24</td>
</tr>
<tr>
<td>2</td>
<td>1.47</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>3.24</td>
</tr>
<tr>
<td>2</td>
<td>-2.50</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2.74</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>2.50</td>
<td>0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.79</td>
</tr>
<tr>
<td>MEMBER</td>
<td>LOAD</td>
<td>JT</td>
<td>AXIAL</td>
<td>SHEAR-Y</td>
<td>SHEAR-Z</td>
<td>TORSION</td>
<td>MOM-Y</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
<td>----</td>
<td>-------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>0.11</td>
<td>-0.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-0.11</td>
<td>0.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.11</td>
<td>0.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-0.11</td>
<td>-0.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>6.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-6.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>4</td>
<td>0.15</td>
<td>0.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-0.15</td>
<td>-0.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>-6.58</td>
<td>0.24</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>6.58</td>
<td>0.24</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>-2.50</td>
<td>0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>2.50</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>5</td>
<td>6.90</td>
<td>0.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>-6.90</td>
<td>0.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>-23.33</td>
<td>-0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>23.33</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>-1.47</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1.47</td>
<td>0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>29.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-29.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>3</td>
<td>11.60</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-11.60</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>4.51</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-4.51</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>2</td>
<td>-9.55</td>
<td>-0.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>9.55</td>
<td>0.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-9.55</td>
<td>0.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>9.55</td>
<td>0.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>8.98</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-8.98</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

51. FINISH
EXAMPLE FOR COMPRESSION-ONLY MEMBERS

*********** END OF THE STAAD.Pro RUN ***********
**** DATE= APR 14,2019   TIME= 22:56:38 ****
EX. US-26 Modeling a Rigid Diaphragm Using Master-Slave

The structure in this example is a building consisting of member columns as well as floors made up of beam members and plate elements. Using the master-slave command, the floors are specified to be rigid diaphragms for in-plane actions but flexible for bending actions.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-26 Modeling a Rigid Diaphragm Using Master-Slave.STD when you install the program.
Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The second line is an optional title to identify this project.

**UNITS KIP FT**

Specify units for the following data.

<table>
<thead>
<tr>
<th>JOINT COORD</th>
<th>1 0 0 4 0 48 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>REPEAT 3 24 0 0</td>
<td></td>
</tr>
<tr>
<td>REPEAT ALL 3 0 0 24</td>
<td></td>
</tr>
<tr>
<td>DELETE JOINT 21 25 37 41</td>
<td></td>
</tr>
</tbody>
</table>

The joint numbers and coordinates are specified above. The unwanted joints, created during the generation process used above, are then deleted.

<table>
<thead>
<tr>
<th>MEMBER INCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12</td>
</tr>
<tr>
<td>13 17 18 15 ; 22 29 30 24 ; 25 33 34 27</td>
</tr>
<tr>
<td>34 45 46 36 ; 37 49 50 39 ; 40 53 54 42</td>
</tr>
<tr>
<td>43 57 58 45 ; 46 61 62 48 ; 49 2 6 51</td>
</tr>
<tr>
<td>52 6 10 54 ; 55 10 14 57 ; 58 18 22 60</td>
</tr>
<tr>
<td>61 22 26 63 ; 64 26 30 66 ; 67 34 38 69</td>
</tr>
<tr>
<td>70 38 42 72 ; 73 42 46 75 ; 76 50 54 78</td>
</tr>
<tr>
<td>79 54 58 81 ; 82 58 62 84 ; 85 18 2 87</td>
</tr>
<tr>
<td>88 22 6 90 ; 91 26 10 93 ; 94 30 14 96</td>
</tr>
<tr>
<td>97 34 18 99 ; 100 38 22 102 ; 103 42 26 105</td>
</tr>
<tr>
<td>106 46 30 108 ; 109 50 34 111 ; 112 54 38 114</td>
</tr>
<tr>
<td>115 58 42 117 ; 118 62 46 120</td>
</tr>
</tbody>
</table>

The MEMBER INCIDENCE specification is used for specifying MEMBER connectivities.

<table>
<thead>
<tr>
<th>ELEMENT INCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>152 50 34 38 54 TO 154</td>
</tr>
<tr>
<td>155 54 38 42 58 TO 157</td>
</tr>
<tr>
<td>158 58 42 46 62 TO 160</td>
</tr>
<tr>
<td>161 34 18 22 38 TO 163</td>
</tr>
<tr>
<td>164 38 22 26 42 TO 166</td>
</tr>
<tr>
<td>167 42 26 30 46 TO 169</td>
</tr>
<tr>
<td>170 18 2 6 22 TO 172</td>
</tr>
<tr>
<td>173 22 6 10 26 TO 175</td>
</tr>
<tr>
<td>176 26 10 14 30 TO 178</td>
</tr>
</tbody>
</table>

The ELEMENT INCIDENCE specification is used for specifying plate element connectivities.

<table>
<thead>
<tr>
<th>MEMBER PROPERTIES AMERICAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 15 22 TO 27 34 TO 48 TA ST W14X90</td>
</tr>
<tr>
<td>49 TO 120 TABLE ST W27X84</td>
</tr>
</tbody>
</table>

All members are WIDE FLANGE sections whose properties are obtained from the built in American steel table.

<table>
<thead>
<tr>
<th>ELEMENT PROP</th>
</tr>
</thead>
<tbody>
<tr>
<td>152 TO 178 THICK 0.75</td>
</tr>
</tbody>
</table>

The thickness of the plate elements is specified above.
The **DEFINE MATERIAL** command is used to specify material properties and the **CONSTANT** is used to assign the material to all members. The orientation of some of the members is set using the **BETA** angle command.

The supports at the above mentioned joints are declared as fixed.

The three floors of the structure are specified to act as rigid diaphragms in the ZX plane with the corresponding master joint specified. The associated slave joints in a floor are specified by the **YRANGE** parameter. The floors may still resist out-of-plane bending actions flexibly.

The above data describe a static load case. It consists of joint loads in the global Z direction.
The above data describe a static load case. It consists of joint loads that create a torsional loading on the structure.

**LOADING 3 DEAD LOAD**
**ELEMENT LOAD**
152 TO 178 PRESS GY -1.0

The above data describe a static load case. It consists of plate element pressure on a floor in the negative global Y direction.

**PERFORM ANALYSIS**

The above command instructs the program to proceed with the analysis.

**PRINT JOINT DISP LIST 4 TO 60 BY 8**
**PRINT MEMBER FORCES LIST 116 115**
**PRINT SUPPORT REACTIONS LIST 9 57**

Print displacements at selected joints, then print member forces for two members, then print support reactions at selected joints.

**FINISH**

The STAAD run is terminated.

**Input File**

**STAAD SPACE**
*MODELING RIGID DIAPHRAGMS USING MASTER SLAVE
UNITS KIP FT
JOINT COORD
1 0 0 0 4 0 48 0
REPEAT 3 24 0 0
REPEAT ALL 3 0 0 24
DELETE JOINT 21 25 37 41
MEMBER INCI
1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12
13 17 18 15 ; 22 29 30 24 ; 25 33 34 27
34 45 46 36 ; 37 49 50 39 ; 40 53 54 42
43 57 58 45 ; 46 61 62 48 ; 49 2 6 51
52 6 10 54 ; 55 10 14 57 ; 58 18 22 60
61 22 26 63 ; 64 26 30 66 ; 67 34 38 69
70 38 42 72 ; 73 42 46 75 ; 76 50 54 78
79 54 58 81 ; 82 58 62 84 ; 85 18 2 87
88 2 6 90 ; 91 26 10 93 ; 94 30 14 96
97 34 18 99 ; 100 38 22 102 ; 103 42 26 105
106 46 30 108 ; 109 50 34 111 ; 112 54 38 114
115 58 42 117 ; 118 62 46 120
ELEMENT INCI
152 50 34 38 54 TO 154
155 54 38 42 58 TO 157
158 58 42 46 62 TO 160
161 34 18 22 38 TO 163
164 38 22 26 42 TO 166
167 42 26 30 46 TO 169
170 18 2 6 22 TO 172
173 22 6 10 26 TO 175
176 26 10 14 30 TO 178
MEMBER PROPERTIES AMERICAN
1 TO 15 22 TO 27 34 TO 48 TA ST W14X90
APPLICATION EXAMPLES
EX. American Design Examples

49 TO 120 TABLE ST W27X84
ELEMENT PROP
152 TO 178 THICK 0.75
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3150
E 2916.7
POISSON 0.17
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 1 TO 15 22 TO 27 34 TO 120
BETA 90.0 MEMB 13 14 15 22 TO 27 34 TO 39
MATERIAL CONCRETE MEMB 152 TO 178
SUPPORTS
1 TO 17 BY 4 29 33 45 TO 61 BY 4 FIXED
SLAVE DIA ZX MASTER 22 JOINTS YR 15.0 17.0
SLAVE DIA ZX MASTER 23 JOINTS YR 31.0 33.0
SLAVE DIA ZX MASTER 24 JOINTS YR 47.0 49.0
LOADING 1 LATERAL LOADS
JOINT LOADS
2 3 4 14 15 16 50 51 52 62 63 64 FZ 10.0
6 7 8 10 11 12 18 19 20 30 31 32 FZ 20.0
34 35 36 46 47 48 54 55 56 58 59 60 FZ 20.0
22 23 24 26 27 28 38 39 40 42 43 44 FZ 40.0
LOADING 2 TORSIONAL LOADS
JOINT LOADS
2 3 4 14 15 16 50 51 52 FZ 5.0
14 15 16 62 63 64 FZ 15.0
6 7 8 18 19 20 FZ 10.0
30 11 12 30 31 32 FZ 30.0
34 35 36 54 55 56 FZ 10.0
46 47 48 58 59 60 FZ 30.0
22 23 24 38 39 40 FZ 20.0
26 27 28 42 43 44 FZ 60.0
LOADING 3 DEAD LOAD
ELEMENT LOAD
152 TO 178 PRESS GY -1.0
PERFORM ANALYSIS
PRINT JOINT DISP LIST 4 TO 60 BY 8
PRINT MEMBER FORCES LIST 116 115
PRINT SUPPORT REACTIONS LIST 9 57
FINISH
1. STAAD SPACE
INPUT FILE: US-26 Modeling a Rigid Diaphragm Using Master-Slave.STD
2. *MODELING RIGID DIAPHRAGMS USING MASTER SLAVE
3. UNITS KIP FT
4. JOINT COORD
5. 1 0 0 0 4 0 48 0
6. REPEAT 3 24 0 0
7. REPEAT ALL 3 0 0 24
8. DELETE JOINT 21 25 37 41
9. MEMBER INCI
10. 1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12
11. 13 17 18 15 ; 22 29 30 24 ; 25 33 34 27
12. 34 45 46 36 ; 37 49 50 39 ; 40 53 54 42
13. 43 57 58 45 ; 46 61 62 48 ; 49 2 6 51
14. 52 6 10 54 ; 55 10 14 57 ; 58 18 22 60
15. 61 22 26 63 ; 64 26 30 66 ; 67 34 38 69
16. 70 38 42 72 ; 73 42 46 75 ; 76 50 54 78
17. 79 54 58 81 ; 82 58 62 84 ; 85 18 2 87
18. 88 22 6 90 ; 91 26 10 93 ; 94 30 14 96
19. 97 34 18 99 ; 100 38 22 102 ; 103 42 26 105
20. 106 46 30 108 ; 109 50 34 111 ; 112 54 38 114
21. 115 58 42 117 ; 118 62 46 120
22. ELEMENT INCI
23. 152 50 34 38 54 TO 154
24. 155 54 38 42 58 TO 157
25. 158 58 42 46 62 TO 160
26. 161 34 18 22 38 TO 163
27. 164 38 22 26 42 TO 166
28. 167 42 26 30 46 TO 169
29. 170 18 2 6 22 TO 172
30. 173 22 6 10 26 TO 175
31. 176 26 10 14 30 TO 178
32. MEMBER PROPERTIES AMERICAN
33. 1 TO 15 22 TO 27 34 TO 48 TA ST W14X90
34. 49 TO 120 TABLE ST W27X84
35. ELEMENT PROP
36. 152 TO 178 THICK 0.75
37. UNIT INCHES
38. DEFINE MATERIAL START
   STAAD SPACE -- PAGE NO. 2
   *MODELING RIGID DIAPHRAGMS USING MASTER SLAVE
39. ISOTROPIC STEEL
40. E 29000
Related Links

- [M. To display master nodes](on page 638)

**EX. US-27 Modeling Soil Springs for a Slab on Grade**

This example illustrates the usage of commands necessary to apply the compression only attribute to spring supports for a slab on grade. The spring supports themselves are generated utilizing the built-in support generation facility. The slab is subjected to pressure and overturning loading. A tension/compression only analysis of the structure is performed.
This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-27 Modeling Soil Springs for a Slab on Grade.STD when you install the program.

The numbers shown in the diagram below are the element numbers.

![Diagram](image)

**Figure 481: Example Problem No. 27**

where:

\[
\begin{align*}
H &= 129', h_1 = 8'-6", h_2 = 8', h_3 = 6' \\
V &= 40', v_1 = 6'-6", v_2 = 6', v_3 = 7'-10", v_4 = 4'
\end{align*}
\]

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. An optional title to identify this project is provided in the second line.

**STAAD SPACE SLAB ON GRADE**

* SPRING COMPRESSION EXAMPLE

This structure has to be analyzed for 3 primary load cases. Consequently, the modeling of our problem requires us to define 3 sets of data, with each set containing a load case and an associated analysis command. Also, the supports which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have 2 commands: SET NL and CHANGE. The SET NL command is used above to indicate the total number of primary load cases that the file contains. The CHANGE command will come in later (after the PERFORM ANALYSIS command).

**UNIT FEET KIP**

**JOINT COORDINATES**

<table>
<thead>
<tr>
<th>Joint</th>
<th>X-coordinate</th>
<th>Y-coordinate</th>
<th>Z-coordinate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>40.0</td>
</tr>
<tr>
<td>2</td>
<td>0.0</td>
<td>0.0</td>
<td>36.0</td>
</tr>
<tr>
<td>3</td>
<td>0.0</td>
<td>0.0</td>
<td>28.167</td>
</tr>
<tr>
<td>4</td>
<td>0.0</td>
<td>0.0</td>
<td>20.333</td>
</tr>
<tr>
<td>5</td>
<td>0.0</td>
<td>0.0</td>
<td>12.5</td>
</tr>
<tr>
<td>6</td>
<td>0.0</td>
<td>0.0</td>
<td>6.5</td>
</tr>
<tr>
<td>7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>REPEAT ALL</td>
<td>3 8.5 0.0 0.0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
For joints 1 through 7, the joint number followed by the X, Y and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these 7 joints by 8.5 feet, 3 times. REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

The incidences of element number 1 is defined and this data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first 6 elements is then used to generate the incidences of 96 more elements using the REPEAT command.

The thickness of elements 1 to 102 is specified as 8.0 inches following the command ELEMENT PROPERTIES.

The modulus of elasticity (E) and Poisson's Ratio are specified following the command CONSTANTS.

The above two lines declare the spring supports at nodes 1 to 126 as having the compression-only attribute. The supports themselves are being generated later (see the ELASTIC MAT command which appears later).

The above command is used to instruct STAAD to generate supports with compression-only springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade reaction of the soil is specified as 12 kip/cu.ft. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the "FY" degree of freedom at the joint. Units for length are changed to FEET to facilitate the input of subgrade reaction of soil. See TR.27.3 Automatic Spring Support Generator for Foundations (on page 2517).
The above data describe a static load case. A pressure load of 1.50 kip/ft acting in the negative global Y direction is applied on all the elements.

Tension/compression cases must each be followed by PERFORM ANALYSIS and CHANGE commands. The CHANGE command restores the original structure to prepare it for the analysis for the next primary load case.

Load case 2 consists of several joint loads acting in the negative global Y direction. This is followed by another ANALYSIS command. The CHANGE command restores the original structure once again for the forthcoming load case.
Load case 3 consists of several joint loads acting in the upward direction at one end and downward on the other end to apply an overturning moment that will lift off one end. The CHANGE command is not needed after the last analysis.

A list of joint displacements, element stresses for elements 34 and 67, and support reactions at a list of joints for load case 3, are obtained with the help of the above commands.

The STAAD run is terminated.

**Input File**

**STAAD SPACE SLAB ON GRADE**

* SPRING COMPRESSION EXAMPLE

SET NL 3
UNIT FEET KIP

JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0 6.5
7 0.0 0.0 0.0

REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0

ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7

UNIT INCH

ELEMENT PROPERTIES
1 TO 102 TH 8.0

DEFINE MATERIAL START

ISOTROPIC CONCRETE
E 4000.0
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4

END DEFINE MATERIAL

CONSTANTS
MATERIAL CONCRETE ALL

SPRING COMPRESSION
1 TO 126 KFY
UNIT FEET
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 12.0
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.50
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2 FY -217.
8 9 FY -109.
5 FY -308.7
6 FY -617.4
22 23 FY -410.
29 30 FY -205.
26 FY -542.7
27 FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82 FY -264.2
48 55 76 83 FY -528.3
92 93 FY -205.0
99 100 FY -410.0
103 FY -487.0
104 FY -974.0
113 114 FY -109.0
120 121 FY -217.0
124 FY -273.3
125 FY -546.6
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 3 'COLUMN OVERTURNING LOAD'
ELEMENT LOAD
1 TO 102 PR GY -1.50
JOINT LOADS
1 2 FY -100.
8 9 FY -50.
5 FY -150.7
6 FY -310.4
22 23 FY -205.
29 30 FY -102.
26 FY -271.7
27 FY -542.4
43 44 50 51 71 72 78 79 FY -153.5
47 54 82 FY -132.2
48 55 76 83 FY -264.3
92 93 FY 102.0
99 100 FY 205.0
103 FY 243.0
104 FY 487.0
113 114 FY 54.0
120 121 FY 108.0
124 FY 136.3
125 FY 273.6
PERFORM ANALYSIS PRINT STATICS CHECK
LOAD LIST 3
PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
PRINT ELEMENT STRESSES LIST 34 67
**Application Examples**

**EX. American Design Examples**

---

**PRINT SUPPORT REACTIONS LIST 5 6 12 13**

**FINISH**

**STAAD Output File**

```
1. STAAD SPACE SLAB ON GRADE
2. * SPRING COMPRESSION EXAMPLE
3. SET NL 3
4. UNIT FEET KIP
5. JOINT COORDINATES
6. 1 0.0 0.0 40.0
7. 2 0.0 0.0 36.0
8. 3 0.0 0.0 28.167
9. 4 0.0 0.0 20.333
10. 5 0.0 0.0 12.5
11. 6 0.0 0.0 6.5
12. 7 0.0 0.0 0.0
13. REPEAT ALL 3 8.5 0.0 0.0
14. REPEAT 3 8.0 0.0 0.0
15. REPEAT 5 6.0 0.0 0.0
16. REPEAT 3 8.0 0.0 0.0
17. REPEAT 3 8.5 0.0 0.0
18. ELEMENT INCIDENCES
19. 1 1 8 9 2 TO 6
20. REPEAT 16 6 7
21. UNIT INCH
22. ELEMENT PROPERTIES
23. 1 TO 102 TH 8.0
24. DEFINE MATERIAL START
25. ISOTROPIC CONCRETE
26. E 4000.0
27. POISSON 0.12
28. DENSITY 8.68E-005
29. ALPHA 5E-006
30. DAMP 0.05
31. G 1346.15
32. TYPE CONCRETE
33. STRENGTH FCU 4
34. END DEFINE MATERIAL
35. CONSTANTS
36. MATERIAL CONCRETE ALL
37. SPRING COMPRESSION
38. 1 TO 126 KFY
* SPRING COMPRESSION EXAMPLE
```

---

**STAAD.Pro**

4586

*User Manual*
39. UNIT FEET
40. SUPPORTS
41. 1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 12.0
42. LOAD 1 'WEIGHT OF MAT & EARTH'
43. ELEMENT LOAD
44. 1 TO 102 PR GY -1.50
45. PERFORM ANALYSIS PRINT STATICS CHECK

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS        126  NUMBER OF MEMBERS        0
NUMBER OF PLATES        102  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS    126

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     378
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE
**NOTE-Tension/Compression converged after 1 iterations, Case=    1
STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
'WEIGHT OF MAT & EARTH'
CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.645000019E+02
Y =  0.000000000E+00
Z =  0.200000006E+02

TOTAL APPLIED LOAD     1
***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING     1 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =       -7740.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=      154800.01  MY=           0.00  MZ=     -499230.03

TOTAL REACTION LOAD    1
***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING     1 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =        7740.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=     -154800.01  MY=           0.00  MZ=      499230.03

MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      1)
MAXIMUMS    AT NODE
X =  0.00000E+00       0
Y = -1.50000E+00       1
Z =  0.00000E+00       0
RX=  9.79653E-10     121
RY=  0.00000E+00       0
RZ=  4.32018E-10      45

************ END OF DATA FROM INTERNAL STORAGE ************

46. CHANGE
47. LOAD 2 'COLUMN LOAD-DL+LL'
48. JOINT LOADS
49. 1 2  FY -217.
50. 8 9  FY -109.
51. 5  FY -308.7
52. 6  FY -617.4
53. 22 23 FY -410.
54. 29 30 FY -205.
SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE

55. 26 FY -542.7
56. 27 FY -1085.4
57. 43 50 51 71 72 78 79 FY -307.5
58. 47 54 82 FY -264.2
59. 48 55 76 83 FY -528.3
60. 92 93 FY -205.0
61. 99 100 FY -410.0
62. 103 FY -487.0
63. 104 FY -974.0
64. 113 114 FY -109.0
65. 120 121 FY -217.0
66. 124 FY -273.3
67. 125 FY -546.6

68. PERFORM ANALYSIS PRINT STATICS CHECK

SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE

**NOTE-Tension/Compression converged after 1 iterations, Case= 2
STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 2
'COLUMN LOAD-DL+LL'
CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)

X = 0.633725605E+02
Y = 0.000000000E+00
Z = 0.215722031E+02

TOTAL APPLIED LOAD 2

***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING 2 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -13964.90
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 301253.66 MY= 0.00 MZ= -884991.47

TOTAL REACTION LOAD 2

***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING 2 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 13964.90
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -301253.66 MY= 0.00 MZ= 884991.47

MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING 2)

MAXIMUMS AT NODE
X = 0.00000E+00 0
Y = -1.12956E+01 120
Z = 0.00000E+00 0
RX= 8.11932E-02 99
RY= 0.00000E+00 0
RZ= 1.00569E-01 6

************ END OF DATA FROM INTERNAL STORAGE ************
SLAB ON GRADE -- PAGE NO. 6

* SPRING COMPRESSION EXAMPLE

78. 22 23 FY -205.
79. 29 30 FY -102.
80. 26 FY -271.7
81. 27 FY -542.4
82. 43 44 50 51 71 72 78 79 FY -153.5
83. 47 54 82 FY -132.2
84. 48 55 76 83 FY -264.3
85. 92 93 FY 102.0
86. 99 100 FY 205.0
87. 103 FY 243.0
88. 104 FY 487.0
89. 113 114 FY 54.0
90. 120 121 FY 108.0
91. 124 FY 136.3
92. 125 FY 273.6
93. PERFORM ANALYSIS PRINT STATICS CHECK

SLAB ON GRADE -- PAGE NO. 7

* SPRING COMPRESSION EXAMPLE

*** LOAD CASE 3 -- START ITERATION NO. 2
*** LOAD CASE 3 -- START ITERATION NO. 3
*** LOAD CASE 3 -- START ITERATION NO. 4
*** LOAD CASE 3 -- START ITERATION NO. 5

**NOTE**-Tension/Compression converged after 5 iterations, Case= 3

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 3

'COLUMN OVERTURNING LOAD'
CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
( FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)

X = 0.45460478E+02
Y = 0.00000000E+00
Z = 0.20712741E+02

TOTAL APPLIED LOAD 3

***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING 3 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -10533.10
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -213519.36 MY= 0.00 MZ= -478687.78

TOTAL REACTION LOAD 3

***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING 3 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 10533.10
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -213519.36 MY= 0.00 MZ= 478687.78

MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING 3)

MAXIMUMS AT NODE
X = 0.00000E+00 0
Y = 3.06386E+01 120
Z = 0.00000E+00 0
RX= -1.35247E-01 120
RY= 0.00000E+00 0
RZ= 1.12342E-01 125

************* END OF DATA FROM INTERNAL STORAGE *************

94. LOAD LIST 3
95. PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
JOINT DISPLACE LIST 113
**SLAB ON GRADE**

* SPRING COMPRESSION EXAMPLE

**JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>113</td>
<td>3</td>
<td>0.00000</td>
<td>20.81368</td>
<td>-0.10277</td>
<td>0.00000</td>
<td>0.07382</td>
</tr>
<tr>
<td>114</td>
<td>3</td>
<td>0.00000</td>
<td>15.82565</td>
<td>-0.10134</td>
<td>0.00000</td>
<td>0.06799</td>
</tr>
<tr>
<td>120</td>
<td>3</td>
<td>0.00000</td>
<td>30.63862</td>
<td>0.00000</td>
<td>-0.13525</td>
<td>0.00000</td>
</tr>
<tr>
<td>121</td>
<td>3</td>
<td>0.00000</td>
<td>24.13106</td>
<td>0.00000</td>
<td>-0.12907</td>
<td>0.00000</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

96. PRINT ELEMENT STRESSES LIST 34 67

**ELEMENT STRESSES LIST 34**

* SPRING COMPRESSION EXAMPLE

**ELEMENT STRESSES  FORCE,LENGTH UNITS= KIP  FEET**

<table>
<thead>
<tr>
<th>ELEMENT LOAD</th>
<th>SQX</th>
<th>SQY</th>
<th>MX</th>
<th>MY</th>
<th>MXY</th>
</tr>
</thead>
<tbody>
<tr>
<td>VONT</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>VONB</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SX</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SY</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SX</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**TRESCAT**  **TRESCAB**

| 34 | -4.66 | -6.79 | 2.34 | 7.95 | 5.74 |
| 164.71 | 164.71| 0.00  | 0.00  | 0.00  |
| 172.44 | 172.44|      |      |      |

**TOP : SMAX= 155.71  SMIN= -16.73  TMAX= 86.22  ANGLE= 58.0**

**BOTT: SMAX= 16.73  SMIN= -155.71  TMAX= 86.22  ANGLE= -32.0**

| 67 | 32.82 | 6.55 | -63.47 | 5.72 | 36.48 |
| 1238.45 | 1238.45| 0.00  | 0.00  | 0.00  |
| 1357.37 | 1357.37|      |      |      |

**TOP : SMAX= 288.91  SMIN= -1068.46  TMAX= 678.69  ANGLE= 66.7**

**BOTT: SMAX= 1068.46  SMIN= -288.91  TMAX= 678.69  ANGLE= -23.3**

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

**MAXIMUM PRINCIPAL MINIMUM PRINCIPAL SHEAR VONMISES TRESCA STRESS STRESS STRESS STRESS STRESS**

1.068459E+03  -1.068459E+03  6.786865E+02  1.238454E+03  1.357373E+03

**PLATE NO. 67  67  67  67  67**

**CASE NO.  3  3  3  3  3**

***************END OF ELEMENT FORCES***************

97. PRINT SUPPORT REACTIONS LIST 5 6 12 13

**SUPPORT REACTION LIST 5**

* SPRING COMPRESSION EXAMPLE

**SUPPORT REACTIONS -UNIT KIP  FEET   STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>3</td>
<td>0.00</td>
<td>149.37</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>0.00</td>
<td>170.57</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>12</td>
<td>3</td>
<td>0.00</td>
<td>148.42</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>13</td>
<td>3</td>
<td>0.00</td>
<td>152.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

98. FINISH

**SLAB ON GRADE**

* SPRING COMPRESSION EXAMPLE

******* END OF THE STAAD.Pro RUN *******

**** DATE= APR 14,2019  TIME= 22:56:47 ****

******************************************************************************

* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/*
EX. US-28 Calculation of Modes and Frequencies of a Bridge

This example demonstrates the input required for obtaining the modes and frequencies of the skewed bridge shown in the figure below. The structure consists of piers, pier-cap girders and a deck slab.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-28 Calculation of Modes and Frequencies of a Bridge.std when you install the program.

Figure 482: Example Problem No. 28

STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words forms a title to identify this project.

IGNORE LIST

Further below in this file, we will call element lists in which some element numbers may not actually be present in the structure. We do so because it minimizes the effort involved in fetching the desired elements and reduces
the size of the respective commands. To prevent the program from treating that condition (referring to elements which do not exist) as an error, the above command is required.

UNIT METER KN

The units for the data that follows are specified above.

JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;
11 -1 0 25 16.5 10 0
REPEAT ALL 3 4 0 14

For joints 1 through 6, the joint number followed by the X, Y and Z coordinates are specified first. Next, using the coordinates of joints 11 and 25 as the basis, joints 12 through 24 are generated using linear interpolation.

Following this, using the data of these 21 joints (1 through 6 and 11 through 25), 63 new joints are generated. To achieve this, the X coordinate of these 21 joints is incremented by 4 meters and the Z coordinate is incremented by 14 meters, in 3 successive operations.

The REPEAT ALL command is used for the generation. Details of this command is available in TR.11 Joint Coordinates Specification (on page 2425). The results of the generation may be visually verified using STAAD.Pro's graphical viewing facilities.

MEMBER INCI
1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86

The member connectivity data (joint numbers between which members are connected) is specified for the 24 columns for the structure. The above method, where the member number is followed by the 2 node numbers, is the explicit definition method. No generation is involved here.

101 11 12 114
202 32 33 215
303 53 54 316
404 74 75 417

The member connectivity data is specified for the pier cap beams for the structure. The above method is a combination of explicit definition and generation. For example, member 101 is defined as connected between 11 & 12. Then, by incrementing those nodes by 1 unit at a time (which is the default increment), the incidences of members 102 to 114 are generated. Similarly, we create members 202 to 215, 303 to 316, and, 404 to 417.

DEFINE MESH
A JOINT 11
B JOINT 25
C JOINT 46
D JOINT 32
E JOINT 67
F JOINT 53
G JOINT 88
H JOINT 74

The next step is to generate the deck slab which will be modeled using plate elements. For this, we use a technique called mesh generation. Mesh generation is a process of generating several "child" elements from a "parent" or "super" element. The above set of commands defines the corner nodes of the super-element. Details of the above can be found in Section 5.14 of the Technical Reference manual.
Note that instead of elaborately defining the coordinates of the corner nodes of the super-elements, we have taken advantage of the fact that the coordinates of these joints (A through H) have already been defined or generated earlier. Thus, A is the same as joint 11 while D is the same as joint 32. Alternatively, we could have defined the super-element nodes as A 1 10 0; B 16.5 10 0; C 20.5 10 14; D 3 10 14; etc.

<table>
<thead>
<tr>
<th>GENERATE ELEMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>MESH ABCD 14 12</td>
</tr>
<tr>
<td>MESH DCEF 14 12</td>
</tr>
<tr>
<td>MESH FEGH 14 12</td>
</tr>
</tbody>
</table>

The above lines are the instructions for generating the “child” elements from the super-elements. For example, from the super-element bound by the corners A, B, C and D (which in turn are nodes 11, 25, 46 and 32), we generate a total of 14X12=168 elements, with 14 divisions along the edges AB and CD, and 12 along the edges BC and DA. These are the elements which make up the first span.

Similarly, 168 elements are created for the 2nd span, and another 168 for the 3rd span.

It may be noted here that we have taken great care to ensure that the resulting elements and the piercap beams form a perfect fit. In other words, there is no overlap between the two in a manner that nodes of the beams are at a different point in space than nodes of elements. At every node along their common boundary, plates and beams are properly connected. This is absolutely essential to ensure proper transfer of load and stiffness from beams to plates and vice versa. The tools of the graphical user interface may be used to confirm that beam-plate connectivity is proper for this model.

<table>
<thead>
<tr>
<th>START GROUP DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMBER</td>
</tr>
<tr>
<td>_GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417</td>
</tr>
<tr>
<td>_PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73</td>
</tr>
<tr>
<td>ELEMENT</td>
</tr>
<tr>
<td>_P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471</td>
</tr>
<tr>
<td>_P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555</td>
</tr>
<tr>
<td>_P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639</td>
</tr>
<tr>
<td>_P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737</td>
</tr>
<tr>
<td>_P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807</td>
</tr>
<tr>
<td>_P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905</td>
</tr>
<tr>
<td>END GROUP DEFINITION</td>
</tr>
</tbody>
</table>

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the piercap beams are being grouped to a name called GIRDERS, the pier columns are assigned the name PIERS, and so on. For the deck, a few selected elements are chosen into a few selective groups. The reason is that these elements happen to be right beneath wheels of vehicles whose weight will be used in the frequency calculation.

<table>
<thead>
<tr>
<th>MEMBER PROPERTY</th>
</tr>
</thead>
<tbody>
<tr>
<td>_GIRDERS PRIS YD 0.6 ZD 0.6</td>
</tr>
<tr>
<td>_PIERS PRIS YD 1.0</td>
</tr>
</tbody>
</table>

Member properties are assigned as prismatic rectangular sections for the girders, and prismatic circular sections for the columns.

<table>
<thead>
<tr>
<th>ELEMENT PROPERTY</th>
</tr>
</thead>
<tbody>
<tr>
<td>YRA 9 11 TH 0.375</td>
</tr>
</tbody>
</table>

The plate elements of the deck slab, which happen to be a Y elevation of 10 metres (between a YRANGE of 9 metres and 11 metres) are assigned a thickness of 375 mms.

<table>
<thead>
<tr>
<th>UNIT KNS MMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOTROPIC CONCRETE</td>
</tr>
<tr>
<td>E 21.0</td>
</tr>
</tbody>
</table>
The Modulus of elasticity \((E)\) is set to 21000 N/sq.mm for all members. The keyword \textit{CONSTANTS} has to precede this data. Built-in default value for Poisson’s ratio for concrete is also assigned to \textit{ALL} members and elements.

Following a change of units, density of concrete is specified.

The base nodes of the piers are fully restrained (FIXED supports).

Theoretically, a structure has as many modes of vibration as the number of degrees of freedom in the model. However, the limitations of the mathematical process used in extracting modes may limit the number of modes that can actually be extracted. In a large structure, the extraction process can also be very time consuming. Further, not all modes are of equal importance. (One measure of the importance of modes is the participation factor of that mode.) In many cases, the first few modes may be sufficient to obtain a significant portion of the total dynamic response.

Due to these reasons, in the absence of any explicit instruction, STAAD calculates only the first 6 modes. This is like saying that the command \texttt{CUT OFF MODE SHAPE 6} has been specified. (Versions of STAAD prior to STAAD.Pro 2000 calculated only 3 modes by default).

If the inspection of the first 6 modes reveals that the overall vibration pattern of the structure has not been obtained, one may ask STAAD to compute a larger (or smaller) number of modes with the help of this command. The number that follows this command is the number of modes being requested. In our example, we are asking for 65 modes by specifying \texttt{CUT OFF MODE SHAPE 65}.

<table>
<thead>
<tr>
<th>UNIT</th>
<th>KNS METER</th>
</tr>
</thead>
<tbody>
<tr>
<td>SELFWEIGHT</td>
<td>X 1.0</td>
</tr>
<tr>
<td>SELFWEIGHT</td>
<td>Y 1.0</td>
</tr>
<tr>
<td>SELFWEIGHT</td>
<td>Z 1.0</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>YRA 9 11 PR GX 200</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>YRA 9 11 PR GY 200</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>YRA 9 11 PR GZ 200</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>P1 PR GY 700</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>P2 PR GY 700</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>P3 PR GY 700</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>P4 PR GY 700</td>
</tr>
<tr>
<td>ELEMENT LOAD</td>
<td>P5 PR GY 700</td>
</tr>
</tbody>
</table>
The mathematical method that STAAD uses is called the eigen extraction method. Some information on this is available in G.17.3 Dynamic Analysis (on page 2362). The method involves 2 matrices - the stiffness matrix, and the mass matrix.

The stiffness matrix, usually called the \([K]\) matrix, is assembled using data such as member and element lengths, member and element properties, modulus of elasticity, Poisson’s ratio, member and element releases, member offsets, support information, etc.

For assembling the mass matrix, called the \([M]\) matrix, STAAD uses the load data specified in the load case in which the MODAL CAL REQ command is specified. So, some of the important aspects to bear in mind are:

1. The input you specify is weights, not masses. Internally, STAAD will convert weights to masses by dividing the input by “g”, the acceleration due to gravity.

2. If the structure is declared as a PLANE frame, there are 2 possible directions of vibration - global X, and global Y. If the structure is declared as a SPACE frame, there are 3 possible directions - global X, global Y and global Z. However, this does not guarantee that STAAD will automatically consider the masses for vibration in all the available directions.

   You have control over and are responsible for specifying the directions in which the masses ought to vibrate. In other words, if a weight is not specified along a certain direction, the corresponding degrees of freedom (such as for example, global X at node 34 hypothetically) will not receive a contribution in the mass matrix. The mass matrix is assembled using only the masses from the weights and directions you specify.

   In our example, notice that we are specifying the self-weight along global X, Y and Z directions. Similarly, a 200 kg/sq.m pressure load is also specified along all 3 directions on the deck.

   But for the truck loads, we choose to apply it on just a few elements in the global Y and Z directions only. The reasoning is something like - for the X direction, the mass is not capable of vibrating because the tires allow the truck to roll along X. Remember, this is just a demonstration example, not necessarily what you may want to do.

   The point we want to illustrate is that if a user wants to restrict a certain weight to certain directions only, all he/she has to do is not provide the directions in which those weights cannot vibrate in.

3. As much as possible, provide absolute values for the weights. STAAD is programmed to algebraically add the weights at nodes. So, if some weights are specified as positive numbers and others as negative, the total weight at a given node is the algebraic summation of all the weights in the global directions at that node and the mass is then derived from this algebraic resultant.

   MODAL CALCULATION REQUESTED

This is the command which tells the program that frequencies and modes should be calculated. It is specified inside a load case. In other words, this command accompanies the loads that are to be used in generating the mass matrix.

Frequencies and modes have to be calculated also when dynamic analysis such as response spectrum or time history analysis is carried out. But in such analyses, the MODAL CALCULATION REQUESTED command is not explicitly required. When STAAD encounters the commands for response spectrum (see example 11) and time
history (see examples 16 and 22), it automatically will carry out a frequency extraction without the help of the MODAL.. command.

**PERFORM ANALYSIS**

This initiates the processes which are required to obtain the frequencies. Frequencies, periods and participation factors are automatically reported in the output file when the operation is completed.

**FINISH**

This terminates the STAAD run.

**Input File**

```
STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
IGNORE LIST
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;
11 -1 10 0 25 16.5 10 0
REPEAT ALL 3 4 0 14
MEMBER INCI
1 1 13; 2 2 15; 3 3 17; 4 4 19; 5 5 21; 6 6 23
26 26 34; 27 27 36; 28 28 38; 29 29 40; 30 30 42; 31 31 44
47 47 55; 48 48 57; 49 49 59; 50 50 61; 51 51 63; 52 52 65
68 68 76; 69 69 78; 70 70 80; 71 71 82; 72 72 84; 73 73 86
101 11 12 114
202 32 33 215
303 53 54 316
404 74 75 417
DEFINE MESH
A JOINT 11
B JOINT 25
C JOINT 46
D JOINT 32
E JOINT 67
F JOINT 53
G JOINT 88
H JOINT 74
GENERATE ELEMENT
MESH ABCD 14 12
MESH DCEF 14 12
MESH FEHG 14 12
START GROUP DEFINITION
MEMBER
_GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
_PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
ELEMENT
_P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
_P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
_P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
_P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
_P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
_P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
END GROUP DEFINITION
MEMBER PROPERTY
_GIRDERS PRIS YD 0.6 ZD 0.6
_PIERS PRIS YD 1.0
ELEMENT PROPERTY
```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL

CONSTANTS
MATERIAL CONCRETE ALL

SUPPORTS
1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED

UNIT KGS METER
LOAD 1 FREQUENCY CALCULATION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
* PERMANENT WEIGHTS ON DECK
ELEMENT LOAD
YRA 9 11 PR GX 200
YRA 9 11 PR GY 200
YRA 9 11 PR GZ 200
* VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
ELEMENT LOAD
_P1 PR GY 700
_P2 PR GY 700
_P3 PR GY 700
_P4 PR GY 700
_P5 PR GY 700
_P6 PR GY 700
_P1 PR GZ 700
_P2 PR GZ 700
_P3 PR GZ 700
_P4 PR GZ 700
_P5 PR GZ 700
_P6 PR GZ 700

MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output File

******************************************************************************************
*                            STAAD.Pro CONNECT Edition                                 *
*                                                                             Version 22.01.00.**  *
*                          Proprietary Program of                                     *
*                          Bentley Systems, Inc.                                     *
* Date= APR 14, 2019                                      *
* Time= 22:56:52                                            *
******************************************************************************************
1. STAAD SPACE FREQUENCIES OF VIBRATION OF A SKewed BRIDGE

INPUT FILE: US-28 Calculation of Modes and Frequencies of a Bridge.STD

2. IGNORE LIST

3. UNIT Meter KN

4. JOINT COORDINATES
   1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0
   11 -1 10 25 16.5 10 0

5. REPEAT ALL 3 4 0 14

6. MEMBER INCI
   1 1 13; 2 2 15; 3 3 17; 4 4 19; 5 5 21; 6 6 23
   26 26 34; 27 27 36; 28 28 38; 29 29 40; 30 30 42; 31 31 44
   47 47 55; 48 48 57; 49 49 59; 50 50 61; 51 51 63; 52 52 65
   68 68 76; 69 69 78; 70 70 80; 71 71 82; 72 72 84; 73 73 86

7. DEFINE MESH

8. A JOINT 11

9. B JOINT 25

10. C JOINT 46

11. D JOINT 32

12. E JOINT 67

13. F JOINT 53

14. G JOINT 88

15. H JOINT 74

16. GENERATE ELEMENT

17. MESH ABCD 14 12

18. MESH DCEF 14 12

19. MESH FEGH 14 12

20. START GROUP DEFINITION

21. MEMBER

22. _GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417

23. _PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73

24. ELEMENT

25. _P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471

26. _P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555

27. _P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639

28. _P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737

29. FREQUENCIES OF VIBRATION OF A SKewed BRIDGE

30. END GROUP DEFINITION

31. MEMBER PROPERTY

32. _GIRDERS PRIS YD 0.6 ZD 0.6

33. _PIERS PRIS YD 1.0

34. ELEMENT PROPERTY

35. YRA 9 11 TH 0.375

36. UNIT KNS MMS

37. DEFINE MATERIAL START

38. ISOTROPIC CONCRETE

39. E 21.0

40. POISSON 0.17

41. DENSITY 2.36158E-008

42. ALPHA 5E-006

43. DAMP 0.05
55. G 9.25
56. TYPE CONCRETE
57. STRENGTH FCU 0.0275
58. END DEFINE MATERIAL
59. CONSTANTS
60. MATERIAL CONCRETE ALL
61. SUPPORTS
62. 1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
63. CUT OFF MODE SHAPE 65
64. UNIT KGS METER
65. LOAD 1 FREQUENCY CALCULATION
66. SELFWEIGHT X 1.0
67. SELFWEIGHT Y 1.0
68. SELFWEIGHT Z 1.0
69. * PERMANENT WEIGHTS ON DECK
70. ELEMENT LOAD
71. YRA 9 11 PR GX 200
72. YRA 9 11 PR GY 200
73. YRA 9 11 PR GZ 200
74. * VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
75. ELEMENT LOAD
76. _P1 PR GY 700
77. _P2 PR GY 700
78. _P3 PR GY 700
79. _P4 PR GY 700
80. _P5 PR GY 700
81. _P6 PR GY 700
82. _P1 PR GZ 700
83. _P2 PR GZ 700
84. _P3 PR GZ 700
85. _P4 PR GZ 700
86. _P5 PR GZ 700
87. _P6 PR GZ 700
88. MODAL CALCULATION REQUESTED
89. PERFORM ANALYSIS
   FREQUENCIES OF VIBRATION OF A SKewed BRIDGE -- PAGE NO. 3
   P R O B L E M   S T A T I S T I C S
   -----------------------------------
   NUMBER OF JOINTS        579  NUMBER OF MEMBERS      80
   NUMBER OF PLATES        504  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     24
   Using 64-bit analysis engine.
   SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =    3330
   TOTAL LOAD COMBINATION CASES =     0  SO FAR.
   ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
   ARE IGNORED. ELEM.NO.   101
   ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
   ARE IGNORED. ELEM.NO.   101
   ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
   ARE IGNORED. ELEM.NO.   101
   ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
   ARE IGNORED. ELEM.NO.   101
   ***NOTE: MASSES DEFINED UNDER LOAD# 1 WILL FORM
   THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
   EIGEN METHOD   : SUBSPACE
   -----------------------------------
   NUMBER OF MODES REQUESTED    =  65
   NUMBER OF EXISTING MASSES IN THE MODEL =  1665
   NUMBER OF MODES THAT WILL BE USED    =  65
<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (CYCLES/SEC)</th>
<th>PERIOD (SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.648</td>
<td>0.60681</td>
</tr>
<tr>
<td>2</td>
<td>2.622</td>
<td>0.38132</td>
</tr>
<tr>
<td>3</td>
<td>2.906</td>
<td>0.34409</td>
</tr>
<tr>
<td>4</td>
<td>3.783</td>
<td>0.26436</td>
</tr>
<tr>
<td>5</td>
<td>4.108</td>
<td>0.24345</td>
</tr>
<tr>
<td>6</td>
<td>4.423</td>
<td>0.22608</td>
</tr>
<tr>
<td>7</td>
<td>4.561</td>
<td>0.21927</td>
</tr>
<tr>
<td>8</td>
<td>4.725</td>
<td>0.21162</td>
</tr>
<tr>
<td>9</td>
<td>5.080</td>
<td>0.19684</td>
</tr>
<tr>
<td>10</td>
<td>7.277</td>
<td>0.13742</td>
</tr>
<tr>
<td>11</td>
<td>7.328</td>
<td>0.13647</td>
</tr>
<tr>
<td>12</td>
<td>7.454</td>
<td>0.13416</td>
</tr>
<tr>
<td>13</td>
<td>10.418</td>
<td>0.09599</td>
</tr>
<tr>
<td>14</td>
<td>10.818</td>
<td>0.09244</td>
</tr>
<tr>
<td>15</td>
<td>11.260</td>
<td>0.08881</td>
</tr>
<tr>
<td>16</td>
<td>11.377</td>
<td>0.08790</td>
</tr>
<tr>
<td>17</td>
<td>11.672</td>
<td>0.08567</td>
</tr>
<tr>
<td>18</td>
<td>11.945</td>
<td>0.08372</td>
</tr>
<tr>
<td>19</td>
<td>12.028</td>
<td>0.08314</td>
</tr>
<tr>
<td>20</td>
<td>12.209</td>
<td>0.08191</td>
</tr>
<tr>
<td>21</td>
<td>12.619</td>
<td>0.07925</td>
</tr>
<tr>
<td>22</td>
<td>13.823</td>
<td>0.07234</td>
</tr>
<tr>
<td>23</td>
<td>14.807</td>
<td>0.06754</td>
</tr>
<tr>
<td>24</td>
<td>14.920</td>
<td>0.06702</td>
</tr>
<tr>
<td>25</td>
<td>15.294</td>
<td>0.06539</td>
</tr>
<tr>
<td>26</td>
<td>17.489</td>
<td>0.05718</td>
</tr>
<tr>
<td>27</td>
<td>17.664</td>
<td>0.05661</td>
</tr>
<tr>
<td>28</td>
<td>17.937</td>
<td>0.05575</td>
</tr>
<tr>
<td>29</td>
<td>19.923</td>
<td>0.05019</td>
</tr>
<tr>
<td>30</td>
<td>20.116</td>
<td>0.04971</td>
</tr>
<tr>
<td>31</td>
<td>20.724</td>
<td>0.04825</td>
</tr>
<tr>
<td>32</td>
<td>20.817</td>
<td>0.04804</td>
</tr>
<tr>
<td>33</td>
<td>21.024</td>
<td>0.04756</td>
</tr>
<tr>
<td>34</td>
<td>21.340</td>
<td>0.04686</td>
</tr>
<tr>
<td>35</td>
<td>21.633</td>
<td>0.04623</td>
</tr>
<tr>
<td>36</td>
<td>22.002</td>
<td>0.04545</td>
</tr>
<tr>
<td>37</td>
<td>22.290</td>
<td>0.04486</td>
</tr>
<tr>
<td>38</td>
<td>23.393</td>
<td>0.04275</td>
</tr>
<tr>
<td>39</td>
<td>23.738</td>
<td>0.04213</td>
</tr>
<tr>
<td>40</td>
<td>24.235</td>
<td>0.04126</td>
</tr>
<tr>
<td>41</td>
<td>24.881</td>
<td>0.04019</td>
</tr>
<tr>
<td>42</td>
<td>25.690</td>
<td>0.03893</td>
</tr>
<tr>
<td>43</td>
<td>26.275</td>
<td>0.03886</td>
</tr>
<tr>
<td>44</td>
<td>26.700</td>
<td>0.03745</td>
</tr>
<tr>
<td>45</td>
<td>27.124</td>
<td>0.03687</td>
</tr>
<tr>
<td>46</td>
<td>27.610</td>
<td>0.03622</td>
</tr>
<tr>
<td>47</td>
<td>28.063</td>
<td>0.03563</td>
</tr>
<tr>
<td>48</td>
<td>29.272</td>
<td>0.03416</td>
</tr>
<tr>
<td>49</td>
<td>29.866</td>
<td>0.03348</td>
</tr>
<tr>
<td>50</td>
<td>30.118</td>
<td>0.03320</td>
</tr>
<tr>
<td>51</td>
<td>31.309</td>
<td>0.03194</td>
</tr>
</tbody>
</table>
## Application Examples

### EX. American Design Examples

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>MODAL WEIGHT (MODAL MASS TIMES g) IN KGS</th>
<th>GENERALIZED WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.291934E+02</td>
<td>1.123418E+00</td>
<td>1.185768E+06</td>
<td>1.205497E+06</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.089784E+06</td>
<td>2.961298E+00</td>
<td>2.290601E+02</td>
<td>1.083499E+06</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2.111306E-01</td>
<td>2.767724E+03</td>
<td>2.799804E+00</td>
<td>5.416754E+05</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1.488365E+00</td>
<td>3.916563E+04</td>
<td>6.302102E+00</td>
<td>1.988939E+05</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>2.650657E-01</td>
<td>4.869465E+02</td>
<td>5.916391E+02</td>
<td>1.348431E+05</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>5.553985E+02</td>
<td>7.047738E+02</td>
<td>2.743600E+02</td>
<td>8.887365E+04</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>1.872127E+01</td>
<td>3.523236E+05</td>
<td>3.255370E+00</td>
<td>7.402632E+04</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>5.441681E+00</td>
<td>2.711803E+05</td>
<td>8.530684E+00</td>
<td>7.667142E+04</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>5.838996E+03</td>
<td>1.713501E+03</td>
<td>2.233780E+03</td>
<td>6.923635E+04</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>4.436158E+00</td>
<td>1.508718E+03</td>
<td>4.95104E+00</td>
<td>4.324923E+04</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>4.522846E+00</td>
<td>6.614279E+02</td>
<td>1.24734E+03</td>
<td>4.305570E+04</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>3.968726E+00</td>
<td>7.524730E+01</td>
<td>4.95104E+00</td>
<td>4.324923E+04</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>2.085909E-01</td>
<td>1.075972E+02</td>
<td>1.23213E+02</td>
<td>7.154577E+04</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>2.752974E-02</td>
<td>3.207858E+00</td>
<td>6.876842E+00</td>
<td>7.610765E+04</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>7.726809E+00</td>
<td>1.937186E+01</td>
<td>1.708543E+01</td>
<td>5.083780E+04</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>3.942715E+00</td>
<td>4.95104E+00</td>
<td>4.324923E+04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>8.006146E+00</td>
<td>5.600056E+01</td>
<td>7.610765E+04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>1.289274E+02</td>
<td>2.295488E+00</td>
<td>5.083780E+04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>3.797408E+00</td>
<td>4.203883E+00</td>
<td>4.324923E+04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>9.95856E+00</td>
<td>7.524730E+01</td>
<td>4.324923E+04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>8.006146E+00</td>
<td>2.974809E+00</td>
<td>4.324923E+04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>1.242891E+02</td>
<td>1.727356E+00</td>
<td>4.324923E+04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>6.938985E+00</td>
<td>3.229583E+02</td>
<td>9.429697E-01</td>
<td>3.20262E+04</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>8.086855E+00</td>
<td>3.229583E+02</td>
<td>9.429697E-01</td>
<td>3.20262E+04</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.01</td>
<td>0.00</td>
<td>99.04</td>
<td>0.012</td>
<td>0.000</td>
<td>99.042</td>
</tr>
<tr>
<td>2</td>
<td>99.14</td>
<td>0.00</td>
<td>0.02</td>
<td>99.152</td>
<td>0.000</td>
<td>99.061</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.23</td>
<td>0.00</td>
<td>99.152</td>
<td>0.232</td>
<td>99.061</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>3.27</td>
<td>0.00</td>
<td>99.152</td>
<td>3.503</td>
<td>99.062</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.04</td>
<td>0.05</td>
<td>99.152</td>
<td>3.544</td>
<td>99.111</td>
</tr>
<tr>
<td>6</td>
<td>0.05</td>
<td>0.06</td>
<td>0.02</td>
<td>99.202</td>
<td>3.602</td>
<td>99.134</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>29.43</td>
<td>0.00</td>
<td>99.284</td>
<td>33.030</td>
<td>99.134</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>22.65</td>
<td>0.00</td>
<td>99.285</td>
<td>55.681</td>
<td>99.135</td>
</tr>
<tr>
<td>9</td>
<td>0.53</td>
<td>0.14</td>
<td>0.19</td>
<td>99.736</td>
<td>55.824</td>
<td>99.322</td>
</tr>
<tr>
<td>10</td>
<td>0.08</td>
<td>0.13</td>
<td>0.00</td>
<td>99.736</td>
<td>55.950</td>
<td>99.322</td>
</tr>
<tr>
<td>11</td>
<td>0.00</td>
<td>0.06</td>
<td>0.00</td>
<td>99.737</td>
<td>56.005</td>
<td>99.322</td>
</tr>
<tr>
<td>12</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>99.737</td>
<td>56.040</td>
<td>99.322</td>
</tr>
<tr>
<td>13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.57</td>
<td>99.740</td>
<td>56.040</td>
<td>99.891</td>
</tr>
<tr>
<td>14</td>
<td>0.00</td>
<td>0.01</td>
<td>0.01</td>
<td>99.740</td>
<td>56.051</td>
<td>99.901</td>
</tr>
<tr>
<td>15</td>
<td>0.00</td>
<td>0.06</td>
<td>0.01</td>
<td>99.740</td>
<td>56.411</td>
<td>99.911</td>
</tr>
<tr>
<td>16</td>
<td>0.00</td>
<td>0.01</td>
<td>0.01</td>
<td>99.741</td>
<td>56.417</td>
<td>99.925</td>
</tr>
<tr>
<td>17</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.742</td>
<td>56.417</td>
<td>99.925</td>
</tr>
<tr>
<td>18</td>
<td>0.01</td>
<td>0.00</td>
<td>0.01</td>
<td>99.754</td>
<td>56.418</td>
<td>99.938</td>
</tr>
<tr>
<td>19</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.754</td>
<td>56.421</td>
<td>99.938</td>
</tr>
<tr>
<td>20</td>
<td>0.00</td>
<td>0.01</td>
<td>0.01</td>
<td>99.754</td>
<td>56.427</td>
<td>99.940</td>
</tr>
<tr>
<td>21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.755</td>
<td>56.428</td>
<td>99.940</td>
</tr>
<tr>
<td>22</td>
<td>0.01</td>
<td>0.00</td>
<td>0.01</td>
<td>99.766</td>
<td>56.428</td>
<td>99.940</td>
</tr>
<tr>
<td>23</td>
<td>0.00</td>
<td>0.01</td>
<td>0.01</td>
<td>99.766</td>
<td>56.431</td>
<td>99.947</td>
</tr>
<tr>
<td>24</td>
<td>0.00</td>
<td>0.01</td>
<td>0.01</td>
<td>99.766</td>
<td>56.447</td>
<td>99.948</td>
</tr>
<tr>
<td>25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.01</td>
<td>99.767</td>
<td>56.448</td>
<td>99.965</td>
</tr>
<tr>
<td>26</td>
<td>0.00</td>
<td>0.00</td>
<td>0.02</td>
<td>99.767</td>
<td>56.630</td>
<td>99.965</td>
</tr>
<tr>
<td>27</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>99.768</td>
<td>56.657</td>
<td>99.965</td>
</tr>
<tr>
<td>28</td>
<td>0.00</td>
<td>1.10</td>
<td>0.00</td>
<td>99.768</td>
<td>57.756</td>
<td>99.965</td>
</tr>
<tr>
<td>29</td>
<td>0.06</td>
<td>0.54</td>
<td>0.00</td>
<td>99.829</td>
<td>58.300</td>
<td>99.970</td>
</tr>
<tr>
<td>MODE</td>
<td>X</td>
<td>Y</td>
<td>Z</td>
<td>SUMM-X</td>
<td>SUMM-Y</td>
<td>SUMM-Z</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>------</td>
<td>------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
</tr>
<tr>
<td>57</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>99.860</td>
<td>73.890</td>
<td>99.974</td>
</tr>
<tr>
<td>58</td>
<td>0.00</td>
<td>0.14</td>
<td>0.00</td>
<td>99.860</td>
<td>74.026</td>
<td>99.975</td>
</tr>
<tr>
<td>59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.861</td>
<td>74.027</td>
<td>99.977</td>
</tr>
<tr>
<td>60</td>
<td>0.00</td>
<td>0.04</td>
<td>0.00</td>
<td>99.861</td>
<td>74.070</td>
<td>99.981</td>
</tr>
<tr>
<td>61</td>
<td>0.14</td>
<td>0.00</td>
<td>0.00</td>
<td>99.866</td>
<td>74.212</td>
<td>99.984</td>
</tr>
<tr>
<td>62</td>
<td>0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>99.868</td>
<td>74.476</td>
<td>99.985</td>
</tr>
<tr>
<td>63</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>99.868</td>
<td>74.521</td>
<td>99.985</td>
</tr>
<tr>
<td>64</td>
<td>0.32</td>
<td>0.00</td>
<td>0.00</td>
<td>99.868</td>
<td>74.836</td>
<td>99.985</td>
</tr>
<tr>
<td>65</td>
<td>0.17</td>
<td>0.00</td>
<td>0.00</td>
<td>99.869</td>
<td>75.008</td>
<td>99.987</td>
</tr>
</tbody>
</table>

**90. FINISH**

FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE -- PAGE NO. 9

*********** END OF THE STAAD.Pro RUN ***********

**** DATE= APR 14,2019 TIME= 22:56:55 ****

******************************************************************************
* For technical assistance on STAAD.Pro, please visit *
* *
* Details about additional assistance from *
* Bentley and Partners can be found at program menu *
* Help->Technical Support *
* *
* Copyright (c) 1997-2017 Bentley Systems, Inc. *
* http://www.bentley.com *
******************************************************************************
Understanding the output

After the analysis is complete, look at the output file. (This file can be viewed by selecting the Analysis Output tool in the View group on the Utilities ribbon tab).

i. Mode number and corresponding frequencies and periods

Since we asked for 65 modes, we obtain a report, a portion of which is as shown:

### Table 657: Calculated Frequencies for Load Case 1

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (Cycles/Sec)</th>
<th>Period (Sec)</th>
<th>Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.636</td>
<td>0.61111</td>
<td>1.344E-16</td>
</tr>
<tr>
<td>2</td>
<td>2.602</td>
<td>0.38433</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>3</td>
<td>2.882</td>
<td>0.34695</td>
<td>8.666E-16</td>
</tr>
<tr>
<td>4</td>
<td>3.754</td>
<td>0.26636</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>5</td>
<td>4.076</td>
<td>0.24532</td>
<td>3.466E-16</td>
</tr>
<tr>
<td>6</td>
<td>4.373</td>
<td>0.22870</td>
<td>6.025E-16</td>
</tr>
<tr>
<td>7</td>
<td>4.519</td>
<td>0.22130</td>
<td>5.641E-16</td>
</tr>
<tr>
<td>8</td>
<td>4.683</td>
<td>0.21355</td>
<td>5.253E-16</td>
</tr>
<tr>
<td>9</td>
<td>5.028</td>
<td>0.19889</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>10</td>
<td>7.189</td>
<td>0.13911</td>
<td>8.916E-16</td>
</tr>
<tr>
<td>11</td>
<td>7.238</td>
<td>0.13815</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>12</td>
<td>7.363</td>
<td>0.13582</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

ii. Participation factors in Percentage

### Table 658: Mass Participation Factors in Percent

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>ΣX</th>
<th>ΣY</th>
<th>ΣZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.01</td>
<td>0.00</td>
<td>99.04</td>
<td>0.012</td>
<td>0.000</td>
<td>99.042</td>
</tr>
<tr>
<td>2</td>
<td>99.14</td>
<td>0.00</td>
<td>0.02</td>
<td>99.151</td>
<td>0.000</td>
<td>99.061</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.23</td>
<td>0.00</td>
<td>99.151</td>
<td>0.229</td>
<td>99.062</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>3.27</td>
<td>0.00</td>
<td>99.151</td>
<td>3.496</td>
<td>99.062</td>
</tr>
</tbody>
</table>
In the explanation earlier for the **CUT OFF MODE** command, we said that one measure of the importance of a mode is the participation factor of that mode. We can see from the above report that for vibration along Z direction, the first mode has a 99.04 percent participation. It is also apparent that the 7th mode is primarily a Y direction mode with a 26.42% participation along Y and 0 in X and Z.

The **ΣX**, **ΣY** and **ΣZ** columns show the cumulative value of the participation of all the modes up to and including a given mode (Corresponding to the SUMM-X, SUMM-Y, and SUMM-Z reported in the output, respectively). One can infer from those terms that if one is interested in 95% participation along X, the first 2 modes are sufficient.

But for the Y direction, even with 10 modes, we barely obtained 60%. The reason for this can be understood by an examination of the nature of the structure. The deck slab is capable of vibrating in several low energy and primarily vertical direction modes. The out-of-plane flexible nature of the slab enables it to vibrate in a manner resembling a series of wave like curves. Masses on either side of the equilibrium point have opposing eigenvector values leading to a lot of cancellation of the contribution from the respective masses. Localized modes, where small pockets in the structure undergo flutter due to their relative weak stiffness compared to the rest of the model, also result in small participation factors.

**iii.** After the analysis is completed, select Post-processing from the mode menu. This screen contains facilities for graphically examining the shape of the mode in static and animated views. The Dynamics page on the left side of the screen is available for viewing the shape of the mode statically. The Animation option of the Results menu can be used for animating the mode. The mode number can be selected from the **Loads and Results** tab of the **Diagrams** dialog box which opens when the Animation option is chosen. The size to which the mode is drawn is controlled using the **Scales** tab of the **Diagrams** dialog box.

**Related Links**
- **M. To calculate the structure frequency** (on page 834)
- **TR.34.2 Modal Calculation Command** (on page 2791)

**EX. US-29 Time History Analysis of a Frame for Seismic Loads**

Analysis and design of a structure for seismic loads is demonstrated in this example. In this model, static load cases are solved along with the seismic load case. For the seismic case, the maximum values of displacements,
forces and reactions are obtained. The results of the dynamic case are combined with those of the static cases and steel design is performed on the combined cases.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-29 Time History Analysis of a Frame for Seismic Loads.std when you install the program.

Figure 483: Example Problem No. 29

Actual input is shown in bold lettering followed by explanation.

STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words form a title to identify this project.

UNIT METER KNS

The units for the data that follows are specified above.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0</td>
</tr>
<tr>
<td>REPEAT ALL 1 9.5 0 0</td>
</tr>
<tr>
<td>REPEAT ALL 1 0 0 3</td>
</tr>
<tr>
<td>17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0</td>
</tr>
<tr>
<td>REPEAT ALL 1 0 0 3</td>
</tr>
</tbody>
</table>

For joints 1 through 4, the joint number is followed by the X, Y and Z coordinates as specified above. The coordinates of these joints are used as a basis for generating 12 more joints by incrementing the X & Z coordinates by specific amounts. REPEAT ALL commands are used for the generation. Details of these commands are available in Section 5.11 of the Technical Reference manual. Following this, another round of
explicit definition (joints 17, 18 & 19) and generation (20, 21 & 22) is carried out. The results of the generation may be visually verified using graphical view features in STAAD.Pro.

MEMBER INCIDENCES
1 1 2 3
REPEAT 1 3 4
7 9 10 9
10 13 14 12
13 4 17; 14 17 18; 15 18 19; 16 19 8
17 12 20; 18 20 21; 19 21 22; 20 22 16
21 2 10; 22 4 12; 23 6 14
24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21

A mixture of explicit definition and generation of member connectivity data (joint numbers between which members are connected) is used to generate 29 members for the structure.

START GROUP DEFINITION
MEMBER
_Vertical 1 TO 12
_Xbeam 13 TO 20
_Zbeam 21 TO 24 29
_Brace 25 TO 28
END GROUP DEFINITION

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the columns are being grouped to a name called VERTICAL, the beams running along the X direction are assigned the name XBEAM, and so on.

MEMBER PROPERTIES CANADIAN
_Vertical TA ST W310X97
_Xbeam TA ST W250X39
_Zbeam TA ST C200X17
_Brace TA ST L152X152X13

Member properties are assigned from the Canadian steel table. The members which receive these properties are those embedded within the respective group names. The benefit of using the group name is apparent here. Just from the looks of the command, we can understand that the diagonal braces are being assigned a single angle. The alternative, which would be

25 TO 28 TA ST L152X152X13

would have required us to go to the graphical tools to get a sense of what members 25 to 28 are.

UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 200
POISSON 0.3
DENSITY 7.8e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 180 MEMB 21 22
The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. The BETA angle for the channels along the left edge is set to 180 so their legs point toward the interior of the structure.

<table>
<thead>
<tr>
<th>SUPPORTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 5 9 13 PINNED</td>
</tr>
</tbody>
</table>

The bottom ends of the columns of the platform are pinned supported.

| CUT OFF MODE SHAPE | 30 |

The above command is a critical command if you want to override the default number of modes computed and used in a dynamic analysis. The default, which is 6, may not always be sufficient to capture a significant portion of the structural response in a response spectrum or time history analysis, and hence the need to override the default. This command is explained in Section 5.30 of the Technical Reference manual.

| UNIT METER DEFINE TIME HISTORY TYPE 1 ACCELERATION READ EQDATA.TXT ARRIVAL TIME 0.0 DAMPING 0.05 |

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the earthquake (ground acceleration) are provided.

Each data set is individually identified by the number that follows the TYPE command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word FORCE that follows the TYPE 1 command signifies that this data set is for a ground acceleration. (If you want to specify a forcing function, the keyword FORCE must be used instead.)

Notice the expression READ EQDATA.TXT. It means that we have chosen to specify the time vs. ground acceleration data in the file called EQDATA.TXT. That file must reside in the same folder as the one in which the data file for this structure resides. As explained in the small examples shown in TR.31.4 Definition of Time History Load (on page 2630), the EQDATA.TXT file is a simple text file containing several pairs of time-acceleration data. A sample portion of that file is as shown below.

| 0.0000 | 0.006300 |
| 0.0200 | 0.003640 |
| 0.0400 | 0.000990 |
| 0.0600 | 0.004280 |
| 0.0800 | 0.007580 |
| 0.1000 | 0.010870 |

While it may not be apparent from the above numbers, it may also be noted that the geological data for the site the building sits on indicate that the above acceleration values are a fraction of "g", the acceleration due to gravity. Thus, for example, at 0.02 seconds, the acceleration is 0.00364 multiplied by 9.806 m/sec² (or 0.00364 multiplied by 32.2 ft/sec²). Consequently, the burden of informing the program that the values need to be multiplied by "g" is upon us, and we shall do so at a later step.

The arrival time value indicates the relative value of time at which the earthquake begins to act upon the structure. We have chosen 0.0, as there is no other dynamic load on the structure from the relative time standpoint. The modal damping ratio for all the modes is set to 0.05.

| LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY SELFWEIGHT Y -1.0 |
The above data describe a static load case. The selfweight of the structure is acting in the negative global Y direction.

Load case 2 is also a static load case. At the Y=7.0m elevation, our structure has a floor slab. But, it is a non-structural entity which, though capable of carrying the loads acting on itself, is not meant to be an integral part of the framing system. It merely transmits the load to the beam-column grid.

There are uniform area loads on the floor (think of the load as wooden pallets supporting boxes of paper). Since the slab is not part of the structural model, how do we tell the program to transmit the imposed load from the slab to the beams without manually converting them to distributed beam loads ourselves? That is where the floor load utility comes in handy. It is a facility where we specify the load as a pressure, and the program converts the pressure to individual beam loads. Thus, the input required is very simple - load intensity in the form of pressure, and the region of the structure in terms of X, Y and Z coordinates in space, of the area over which the pressure acts.

In the process of converting the pressure to beam loads, STAAD will consider the empty space between criss-crossing beams (in plan view) to be panels, similar to the squares of a chess board. The load on each panel is then transferred to beams surrounding the panel, using a triangular or trapezoidal load distribution method.

Load case 3 is the dynamic load case, the one which contains the second part of the instruction set for a dynamic analysis to be performed. The data here are

a. loads which will yield the mass values which will populate the mass matrix
b. the directions of the loads, which will yield the degree of freedom numbers of the mass matrix for being populated.

Thus, the selfweight, as well as the imposed loads on the non-structural slab are to be considered as participating in the vibration along all the global directions.

The above command too is part of load case 3. Here we say that the seismic force, whose characteristics are defined by the TYPE 1 time history input data, acting at arrival time 1, is to be applied along the X direction. We mentioned earlier that the acceleration input data was specified as a fraction of “g”. The number 9.806 indicates the value which the acceleration data, as read from EQDATA.TXT are to be factored by before they are used.

In a time history analysis, the member forces FX thru MZ each have a value for every time step. If there are 1000 time steps, there will be 1000 values of FX, 1000 for FY etc. for that load case. Not all of them can be used in a further calculation like a steel or concrete design. However, the maximum from among those time steps is available. If we want to do a design, one way to make sure that the structure is not under-designed is to create 2 load combination cases involving the dynamic case, a positive combination, and a negative combination.
That is what is being done above. Load combination case no. 11 consists of the sum of the static load cases (1 & 2) with the positive direction of the dynamic load case (3). Load combination case no. 12 consists of the sum of the static load cases (1 & 2) with the negative direction of the dynamic load case (3). The user has discretion on what load factors to use with these combinations. We have chosen the factors to be 1.0.

**PERFORM ANALYSIS**

The above is the instruction to perform the analysis related calculations. That means, computing nodal displacements, support reactions, etc.

**PRINT ANALYSIS RESULTS**

The above command is an instruction to the program to produce a report of the joint displacements, support reactions and member end forces in the output file. As mentioned earlier, for the dynamic case, these will be just the maximum values, not the ones generated for every time step. If you want to see the results for each time step, you may do so by using STAAD's Post-processing facilities.

```
LOAD LIST 11 12
PARAMETER
CODE CANADIAN
CHECK CODE ALL
```

A steel design code check is done according to the Canadian code for load cases 11 and 12.

**FINISH**

**Input File**

```
STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
UNIT METER KNS
JOINT COORDINATES
1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
REPEAT ALL 1 9.5 0 0
REPEAT ALL 1 0 0 3
17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
REPEAT ALL 1 0 0 3
MEMBER INCIDENCES
1 1 2 3
REPEAT 1 3 4
7 9 10 9
10 13 14 12
13 4 17; 14 17 18; 15 18 19; 16 19 8
17 12 20; 18 20 21; 19 21 22; 20 22 16
21 2 10; 22 4 12; 23 6 14
24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
START GROUP DEFINITION
MEMBER
__VERTICAL 1 TO 12
__XBEAM 13 TO 20
__ZBEAM 21 TO 24 29
__BRACE 25 TO 28
END GROUP DEFINITION
MEMBER PROPERTIES CANADIAN
__VERTICAL TA ST W310X97
__XBEAM TA ST W250X39
__ZBEAM TA ST C200X17
__BRACE TA ST L152X152X13
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
```
E 200
POISSON 0.3
DENSITY 7.8e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 180 MEMB 21 22
SUPPORTS
1 5 9 13 PINNED
CUT OFF MODE SHAPE 30
UNIT KGS METER
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05
LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
SELFWEIGHT Y -1.0
LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY
FLOOR LOAD
YRA 6.9 7.1 FLOAD -500
LOAD 3 DYNAMIC LOAD
* MASSES
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
FLOOR LOAD
YRANGE 6.9 7.1 FLOAD 500 GX
YRANGE 6.9 7.1 FLOAD 500 GY
YRANGE 6.9 7.1 FLOAD 500 GZ
GROUND MOTION X 1 1 9.806
LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
1.0 2 1.0 3 1.0
LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
1.0 2 1.0 3 -1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 11 12
PARAMETER
CODE CANADIAN
CHECK CODE ALL
FINISH

STAAD Output File

****************************************************************************************** PAGE NO. 1
*                                                                                   *
* STAAD.Pro CONNECT Edition                                                      *
* Version 22.01.00.**                                                              *
* Proprietary Program of                                                          *
* Bentley Systems, Inc.                                                           *
* Date= APR 14, 2019                                                             *
1. STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
INPUT FILE: US-29 Time History Analysis of a Frame for Seismic Loads.STD
2. UNIT METER KNS
3. JOINT COORDINATES
4. 1 0 0 0; 2 0 3.5 0; 3 0 5.3 0; 4 0 7 0
5. REPEAT ALL 1 9.5 0 0
6. REPEAT ALL 1 0 0 3
7. 17 1.8 7 0; 18 4.6 7 0; 19 7.6 7 0
8. REPEAT ALL 1 0 0 3
9. MEMBER INCIDENCES
10. 1 1 2 3
11. REPEAT 1 3 4
12. 7 9 10 9
13. 10 13 14 12
14. 13 4 17; 14 17 18; 15 18 19; 16 19 8
15. 17 12 20; 18 20 21; 19 21 22; 20 22 16
16. 21 2 10; 22 4 12; 23 6 14
17. 24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
18. START GROUP DEFINITION
19. MEMBER
20. _VERTICAL 1 TO 12
21. _XBEAM 13 TO 20
22. _ZBEAM 21 TO 24 29
23. _BRACE 25 TO 28
24. END GROUP DEFINITION
25. MEMBER PROPERTIES CANADIAN
26. _VERTICAL TA ST W310X97
27. _XBEAM TA ST W250X39
28. _ZBEAM TA ST C200X17
29. _BRACE TA ST L152X152X13
30. UNIT KNS MMS
31. DEFINE MATERIAL START
32. ISOTROPIC STEEL
33. E 200
34. POISSON 0.3
35. DENSITY 7.8E-008
36. ALPHA 6E-006
37. DAMP 0.03
38. TYPE STEEL
39. DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 2
40. END DEFINE MATERIAL
41. CONSTANTS
42. MATERIAL STEEL ALL
43. BETA 180 MEMB 21 22
44. SUPPORTS
45. 1 5 9 13 PINNED
46. CUT OFF MODE SHAPE 30
47. UNIT KGS METER
48. DEFINE TIME HISTORY
49. TYPE 1 ACCELERATION
50. READ EQDATA.TXT
51. ARRIVAL TIME
52. 0.0
**NOTE** about Floor/OneWay Loads/Weights.
Please note that depending on the shape of the floor you may have to break up the FLOOR/ONEWAY LOAD into multiple commands. For details please refer to Technical Reference Manual Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.

59. LOAD 3 DYNAMIC LOAD

60. * MASSES
   61. SELFWEIGHT X 1.0
   62. SELFWEIGHT Y 1.0
   63. SELFWEIGHT Z 1.0
64. FLOOR LOAD
65. YRANGE 6.9 7.1 FLOAD 500 GX
66. YRANGE 6.9 7.1 FLOAD 500 GY
67. YRANGE 6.9 7.1 FLOAD 500 GZ
68. GROUND MOTION X 1 1 9.806
69. LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
   70. 1 1.0 2 1.0 3 1.0
71. LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
   72. 1 1.0 2 1.0 3 -1.0
73. PERFORM ANALYSIS

DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 3

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS 22  NUMBER OF MEMBERS 29
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 4

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 3, TOTAL DEGREES OF FREEDOM = 120
TOTAL LOAD COMBINATION CASES = 2 SO FAR.
***NOTE: MASSES DEFINED UNDER LOAD# 3 WILL FORM THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.

EIGEN METHOD : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED = 30
NUMBER OF EXISTING MASSES IN THE MODEL = 54
NUMBER OF MODES THAT WILL BE USED = 30

*** EIGENSOLUTION : ADVANCED METHOD ***

DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 4
CALCULATED FREQUENCIES FOR LOAD CASE 3

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.693</td>
<td>1.44295</td>
</tr>
<tr>
<td>2</td>
<td>1.215</td>
<td>0.82296</td>
</tr>
<tr>
<td>3</td>
<td>1.365</td>
<td>0.73265</td>
</tr>
<tr>
<td>4</td>
<td>1.561</td>
<td>0.64059</td>
</tr>
<tr>
<td>5</td>
<td>2.077</td>
<td>0.48142</td>
</tr>
<tr>
<td>6</td>
<td>3.044</td>
<td>0.32846</td>
</tr>
<tr>
<td>7</td>
<td>4.217</td>
<td>0.23712</td>
</tr>
<tr>
<td>8</td>
<td>4.273</td>
<td>0.23404</td>
</tr>
<tr>
<td>9</td>
<td>5.538</td>
<td>0.18058</td>
</tr>
<tr>
<td>10</td>
<td>5.543</td>
<td>0.18039</td>
</tr>
<tr>
<td>11</td>
<td>5.728</td>
<td>0.17457</td>
</tr>
</tbody>
</table>
## MODAL WEIGHT (MODAL MASS TIMES g) IN KGS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.3300908E-16</td>
<td>3.856561E-17</td>
<td>1.501941E+04</td>
<td>8.845560E+03</td>
</tr>
<tr>
<td>2</td>
<td>1.725302E+04</td>
<td>4.369968E-02</td>
<td>2.140145E-16</td>
<td>1.677679E+04</td>
</tr>
<tr>
<td>3</td>
<td>2.909841E-11</td>
<td>5.820389E-16</td>
<td>8.419906E-01</td>
<td>1.266359E+04</td>
</tr>
<tr>
<td>4</td>
<td>8.919546E-12</td>
<td>1.342864E-15</td>
<td>4.822829E+00</td>
<td>2.596549E+04</td>
</tr>
<tr>
<td>5</td>
<td>5.385775E-16</td>
<td>1.266211E-18</td>
<td>2.369262E+03</td>
<td>6.440201E+03</td>
</tr>
<tr>
<td>6</td>
<td>2.267313E-29</td>
<td>3.707073E-30</td>
<td>8.107877E-14</td>
<td>5.200768E+03</td>
</tr>
<tr>
<td>7</td>
<td>6.805920E-30</td>
<td>3.131805E-29</td>
<td>5.977653E-13</td>
<td>4.004403E+03</td>
</tr>
<tr>
<td>8</td>
<td>3.523092E-16</td>
<td>2.639318E-16</td>
<td>1.923307E+00</td>
<td>8.821168E+03</td>
</tr>
<tr>
<td>9</td>
<td>6.201204E-03</td>
<td>9.100303E+03</td>
<td>1.326257E-10</td>
<td>6.018268E+03</td>
</tr>
<tr>
<td>10</td>
<td>3.796626E-12</td>
<td>5.397674E-06</td>
<td>2.331646E-01</td>
<td>6.033971E+03</td>
</tr>
<tr>
<td>11</td>
<td>3.658195E-17</td>
<td>1.386745E-11</td>
<td>4.085088E+01</td>
<td>8.734127E+03</td>
</tr>
<tr>
<td>12</td>
<td>3.013838E+02</td>
<td>4.352953E+01</td>
<td>5.141852E-13</td>
<td>6.235738E+03</td>
</tr>
</tbody>
</table>

## DYNAMIC ANALYSIS FOR SEISMIC LOADS

### MASS PARTICIPATION FACTORS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>85.32</td>
<td>0.00</td>
<td>0.00</td>
<td>85.32</td>
</tr>
</tbody>
</table>

---

**Application Examples**

**EX. American Design Examples**

STAAD.Pro 4614 User Manual
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Actual Modal Damping Used in Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.05000000</td>
</tr>
<tr>
<td>2</td>
<td>0.05000000</td>
</tr>
<tr>
<td>3</td>
<td>0.05000000</td>
</tr>
<tr>
<td>4</td>
<td>0.05000000</td>
</tr>
<tr>
<td>5</td>
<td>0.05000000</td>
</tr>
<tr>
<td>6</td>
<td>0.05000000</td>
</tr>
<tr>
<td>7</td>
<td>0.05000000</td>
</tr>
<tr>
<td>8</td>
<td>0.05000000</td>
</tr>
<tr>
<td>9</td>
<td>0.05000000</td>
</tr>
<tr>
<td>10</td>
<td>0.05000000</td>
</tr>
<tr>
<td>11</td>
<td>0.05000000</td>
</tr>
<tr>
<td>12</td>
<td>0.05000000</td>
</tr>
<tr>
<td>13</td>
<td>0.05000000</td>
</tr>
<tr>
<td>14</td>
<td>0.05000000</td>
</tr>
<tr>
<td>15</td>
<td>0.05000000</td>
</tr>
<tr>
<td>16</td>
<td>0.05000000</td>
</tr>
</tbody>
</table>

**Dynamic Analysis for Seismic Loads**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Damping</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>0.05000000</td>
</tr>
<tr>
<td>18</td>
<td>0.05000000</td>
</tr>
<tr>
<td>19</td>
<td>0.05000000</td>
</tr>
<tr>
<td>20</td>
<td>0.05000000</td>
</tr>
<tr>
<td>21</td>
<td>0.05000000</td>
</tr>
<tr>
<td>22</td>
<td>0.05000000</td>
</tr>
<tr>
<td>23</td>
<td>0.05000000</td>
</tr>
<tr>
<td>24</td>
<td>0.05000000</td>
</tr>
<tr>
<td>25</td>
<td>0.05000000</td>
</tr>
</tbody>
</table>
26 0.05000000
27 0.05000000
28 0.05000000
29 0.05000000
30 0.05000000

TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED = 30
TIME DURATION OF TIME HISTORY ANALYSIS = 31.160 SECONDS
NUMBER OF TIME STEPS IN THE SOLUTION PROCESS = 22435

74. PRINT ANALYSIS RESULTS

BASE SHEAR UNITS ARE -- KGS METE
MAXIMUM BASE SHEAR X= -9.498281E+03 Y= -5.285205E+01 Z= 4.260048E-06
AT TIMES 5.809722 2.445833 2.766667

ANALYSIS RESULTS

DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 8

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0015</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0173</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0157</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0190</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-0.0290</td>
<td>-0.0012</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>-0.4146</td>
<td>-0.0050</td>
<td>-0.0004</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>0.0004</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5.7319</td>
<td>0.0046</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0142</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>5.2883</td>
<td>-0.0015</td>
<td>-0.0005</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>-0.0142</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-6.1755</td>
<td>-0.0108</td>
<td>-0.0005</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>0.0138</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-0.0249</td>
<td>-0.0016</td>
<td>-0.0002</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>2</td>
<td>-0.3581</td>
<td>-0.0075</td>
<td>-0.0122</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>7.9529</td>
<td>0.0070</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0101</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>7.5699</td>
<td>-0.0021</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0113</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-8.3358</td>
<td>-0.0161</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0088</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-0.0830</td>
<td>-0.0016</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>2</td>
<td>-0.8451</td>
<td>-0.0076</td>
<td>0.0004</td>
<td>0.0002</td>
<td>-0.0000</td>
<td>-0.0020</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9.3683</td>
<td>0.0023</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0083</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>9.3201</td>
<td>-0.0069</td>
<td>0.0004</td>
<td>0.0002</td>
<td>0.0000</td>
<td>-0.104</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-9.4164</td>
<td>-0.0115</td>
<td>0.0004</td>
<td>0.0002</td>
<td>-0.0000</td>
<td>0.0062</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0014</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0174</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0189</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0159</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.0261</td>
<td>-0.0012</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.3723</td>
<td>-0.0050</td>
<td>-0.0004</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>-0.0002</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5.7484</td>
<td>-0.0046</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0142</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>6.1468</td>
<td>-0.0108</td>
<td>-0.0005</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>-0.144</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-5.3500</td>
<td>0.0015</td>
<td>0.0005</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0139</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>0.0206</td>
<td>-0.0016</td>
<td>-0.0002</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.2940</td>
<td>-0.0075</td>
<td>-0.0122</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0013</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>7.9667</td>
<td>-0.0070</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0100</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>8.2813</td>
<td>-0.0161</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>0.0001</td>
<td>-0.0086</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-7.6521</td>
<td>-0.0021</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>0.0001</td>
<td>0.0114</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>-0.0827</td>
<td>-0.0016</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>-0.3987</td>
<td>-0.0077</td>
<td>0.0004</td>
<td>0.0002</td>
<td>0.0000</td>
<td>0.0021</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9.3696</td>
<td>-0.0026</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0082</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>9.3271</td>
<td>-0.0119</td>
<td>0.0004</td>
<td>0.0002</td>
<td>0.0000</td>
<td>-0.0060</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-9.4121</td>
<td>-0.0068</td>
<td>0.0004</td>
<td>0.0002</td>
<td>0.0000</td>
<td>0.0105</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rotan</th>
<th>Y-Rotan</th>
<th>Z-Rotan</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0015</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0173</td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0157</td>
</tr>
<tr>
<td>12</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0190</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>-0.0290</td>
<td>-0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 9

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE

---

**STAAD.Pro**

4617

User Manual
### Joint Load

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rotan</th>
<th>Y-Rotan</th>
<th>Z-Rotan</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>9.3514</td>
<td>1.4648</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0031</td>
</tr>
<tr>
<td>11</td>
<td>9.3032</td>
<td>1.0237</td>
<td>0.0009</td>
<td>0.0001</td>
<td>-0.0000</td>
<td>-0.0004</td>
<td>0.0000</td>
</tr>
<tr>
<td>12</td>
<td>-9.3995</td>
<td>-1.9606</td>
<td>0.0009</td>
<td>0.0001</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0057</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>-0.0270</td>
<td>-0.0267</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
</tr>
<tr>
<td>2</td>
<td>-0.0400</td>
<td>-0.3687</td>
<td>-0.0008</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>-0.0025</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>9.3513</td>
<td>-1.4019</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0033</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>9.3086</td>
<td>-1.7973</td>
<td>0.0009</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>-0.0059</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-9.3941</td>
<td>1.0065</td>
<td>-0.0008</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>0.0006</td>
<td></td>
</tr>
</tbody>
</table>

### Support Reactions

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>61.39</td>
<td>1009.30</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-872.77</td>
<td>3562.50</td>
<td>-19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-2356.64</td>
<td>-3311.14</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-1422.49</td>
<td>1260.65</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>3290.79</td>
<td>7882.94</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-61.39</td>
<td>1009.45</td>
<td>-61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-872.77</td>
<td>3562.50</td>
<td>19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-2392.50</td>
<td>3308.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-3326.65</td>
<td>7880.23</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>1458.35</td>
<td>1263.66</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>61.39</td>
<td>1009.30</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-872.77</td>
<td>3562.50</td>
<td>-19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-2356.64</td>
<td>-3311.14</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-1422.49</td>
<td>1260.65</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>3290.79</td>
<td>7882.94</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>-61.39</td>
<td>1009.45</td>
<td>-61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-872.77</td>
<td>3562.50</td>
<td>19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-2392.50</td>
<td>3308.29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-3326.65</td>
<td>7880.23</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>1458.35</td>
<td>1263.66</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>641.00</td>
<td>-61.39</td>
<td>5.97</td>
<td>0.01</td>
<td>-9.03</td>
<td>214.85</td>
</tr>
<tr>
<td>3</td>
<td>-464.90</td>
<td>-61.39</td>
<td>-5.97</td>
<td>-0.01</td>
<td>-1.72</td>
<td>-124.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>3562.50</td>
<td>-872.77</td>
<td>112.31</td>
<td>0.24</td>
<td>-79.35</td>
<td>3054.69</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-3562.50</td>
<td>872.77</td>
<td>-112.31</td>
<td>-0.24</td>
<td>-124.85</td>
<td>4625.67</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-461.23</td>
<td>872.77</td>
<td>-112.31</td>
<td>-0.24</td>
<td>-124.85</td>
<td>3054.69</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>191.68</td>
<td>-61.39</td>
<td>5.97</td>
<td>-0.01</td>
<td>1.70</td>
<td>310.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-25.36</td>
<td>-61.39</td>
<td>-5.97</td>
<td>-0.01</td>
<td>-11.84</td>
<td>7.31</td>
<td>3054.69</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>3562.50</td>
<td>872.77</td>
<td>112.31</td>
<td>0.24</td>
<td>-79.35</td>
<td>3054.69</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-3562.50</td>
<td>-872.77</td>
<td>-112.31</td>
<td>-0.24</td>
<td>-124.85</td>
<td>4625.67</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-461.23</td>
<td>-872.77</td>
<td>-112.31</td>
<td>-0.24</td>
<td>-124.85</td>
<td>3054.69</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**DYNAMIC ANALYSIS FOR SEISMIC LOADS**

**MEMBER END FORCES**

**STRUCTURE TYPE** = SPACE

---

**All units are -- KGS METE (LOCAL)**

---

### STAAD.Pro 4619 User Manual
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>9</td>
<td>3562.50</td>
<td>-872.77</td>
<td>19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-3562.50</td>
<td>872.77</td>
<td>-19.74</td>
<td>0.00</td>
<td>-69.09</td>
<td>-3054.69</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>-3311.14</td>
<td>2356.64</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>3311.14</td>
<td>-2356.64</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>9</td>
<td>1260.65</td>
<td>1422.49</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-918.24</td>
<td>-1422.49</td>
<td>-18.76</td>
<td>0.00</td>
<td>-65.66</td>
<td>4978.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>9</td>
<td>12304.78</td>
<td>-3290.79</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-7540.53</td>
<td>3290.79</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Member End Forces**

**Structure Type = Space**

---

**All Units Are -- Kgs Mete (Local)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>11</td>
<td>41.65</td>
<td>2832.69</td>
<td>-111.63</td>
<td>0.35</td>
<td>-122.13</td>
<td>4570.39</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-41.65</td>
<td>-2832.69</td>
<td>111.63</td>
<td>-0.35</td>
<td>311.90</td>
<td>245.19</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>11</td>
<td>6886.37</td>
<td>-8000.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-12336.68</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-6886.37</td>
<td>8000.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2793.95</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>11</td>
<td>7119.69</td>
<td>-5881.00</td>
<td>18.76</td>
<td>0.36</td>
<td>-123.83</td>
<td>7456.29</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-6953.38</td>
<td>5881.00</td>
<td>-17.60</td>
<td>-0.36</td>
<td>323.74</td>
<td>2541.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>11</td>
<td>7880.23</td>
<td>3326.65</td>
<td>18.76</td>
<td>0.36</td>
<td>-123.83</td>
<td>17217.07</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-7537.82</td>
<td>-3326.65</td>
<td>-18.76</td>
<td>-0.36</td>
<td>8373.75</td>
<td>3046.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>11</td>
<td>1009.45</td>
<td>61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-667.04</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>3.43</td>
<td>214.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>13</td>
<td>3562.50</td>
<td>872.77</td>
<td>19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-3562.50</td>
<td>-872.77</td>
<td>-19.74</td>
<td>0.00</td>
<td>-69.09</td>
<td>3054.69</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>13</td>
<td>3308.29</td>
<td>2392.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>8373.75</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-3308.29</td>
<td>-2392.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2793.95</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>13</td>
<td>7880.23</td>
<td>3326.65</td>
<td>18.76</td>
<td>0.36</td>
<td>-123.83</td>
<td>7456.29</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-7537.82</td>
<td>-3326.65</td>
<td>-18.76</td>
<td>-0.36</td>
<td>323.74</td>
<td>2541.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>13</td>
<td>1263.66</td>
<td>1458.35</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-921.25</td>
<td>1458.35</td>
<td>-18.76</td>
<td>0.00</td>
<td>-65.65</td>
<td>-5104.22</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>1009.45</td>
<td>61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-667.04</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>3.43</td>
<td>214.85</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**DYNAMIC ANALYSIS FOR SEISMIC LOADS**

**Member End Forces**

**Structure Type = SPACE**

---

**All Units Are -- Kgs Mete (Local)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>11</td>
<td>41.65</td>
<td>2832.69</td>
<td>-111.63</td>
<td>0.35</td>
<td>-122.13</td>
<td>4570.39</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-41.65</td>
<td>-2832.69</td>
<td>111.63</td>
<td>-0.35</td>
<td>311.90</td>
<td>245.19</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>11</td>
<td>6886.37</td>
<td>-8000.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-12336.68</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-6886.37</td>
<td>8000.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2793.95</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>11</td>
<td>7119.69</td>
<td>-5881.00</td>
<td>18.76</td>
<td>0.36</td>
<td>-123.83</td>
<td>7456.29</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-6953.38</td>
<td>5881.00</td>
<td>-17.60</td>
<td>-0.36</td>
<td>323.74</td>
<td>2541.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>11</td>
<td>7880.23</td>
<td>3326.65</td>
<td>18.76</td>
<td>0.36</td>
<td>-123.83</td>
<td>17217.07</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-7537.82</td>
<td>-3326.65</td>
<td>-18.76</td>
<td>-0.36</td>
<td>8373.75</td>
<td>3046.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>1009.45</td>
<td>61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-667.04</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>3.43</td>
<td>214.85</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**STAAD.Pro**

**User Manual**
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>4</td>
<td>6423.46</td>
<td>6364.76</td>
<td>-0.70</td>
<td>0.09</td>
<td>-1.54</td>
<td>2711.06</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>4</td>
<td>-6237.20</td>
<td>12040.72</td>
<td>117.59</td>
<td>0.34</td>
<td>323.75</td>
<td>7651.26</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>186.66</td>
<td>70.96</td>
<td>-0.53</td>
<td>0.01</td>
<td>-0.03</td>
<td>-57.03</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>1570.80</td>
<td>1570.80</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>2793.95</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>4</td>
<td>-6997.09</td>
<td>6997.09</td>
<td>117.59</td>
<td>0.34</td>
<td>323.75</td>
<td>7651.26</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>186.66</td>
<td>70.96</td>
<td>-0.53</td>
<td>0.01</td>
<td>-0.03</td>
<td>-57.03</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>1570.80</td>
<td>1570.80</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>2793.95</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>4</td>
<td>-6997.09</td>
<td>6997.09</td>
<td>117.59</td>
<td>0.34</td>
<td>323.75</td>
<td>7651.26</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>186.66</td>
<td>70.96</td>
<td>-0.53</td>
<td>0.01</td>
<td>-0.03</td>
<td>-57.03</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>1570.80</td>
<td>1570.80</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>2793.95</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>4</td>
<td>-6997.09</td>
<td>6997.09</td>
<td>117.59</td>
<td>0.34</td>
<td>323.75</td>
<td>7651.26</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>186.66</td>
<td>70.96</td>
<td>-0.53</td>
<td>0.01</td>
<td>-0.03</td>
<td>-57.03</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>1570.80</td>
<td>1570.80</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>2793.95</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>4</td>
<td>-6997.09</td>
<td>6997.09</td>
<td>117.59</td>
<td>0.34</td>
<td>323.75</td>
<td>7651.26</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>186.66</td>
<td>70.96</td>
<td>-0.53</td>
<td>0.01</td>
<td>-0.03</td>
<td>-57.03</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>1570.80</td>
<td>1570.80</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>2793.95</td>
<td></td>
</tr>
</tbody>
</table>

#### DYNAMIC ANALYSIS FOR SEISMIC LOADS

---

**Member End Forces**  
**Structure Type = SPACE**

---

**All Units Are -- KGS METE (LOCAL)**

---

**STAAD.Pro User Manual**
<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>Jt</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>12</td>
<td>6423.46</td>
<td>6364.76</td>
<td>0.70</td>
<td>-0.09</td>
<td>0.30</td>
<td>2541.45</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>-6423.46</td>
<td>-5506.82</td>
<td>-0.70</td>
<td>0.09</td>
<td>-1.57</td>
<td>8367.98</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>12</td>
<td>-12462.15</td>
<td>8265.46</td>
<td>-0.70</td>
<td>0.09</td>
<td>-1.57</td>
<td>-10834.23</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>12462.15</td>
<td>-7407.52</td>
<td>-0.70</td>
<td>0.09</td>
<td>-1.57</td>
<td>-10005.50</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>61.39</td>
<td>129.40</td>
<td>0.00</td>
<td>0.03</td>
<td>-0.01</td>
<td>67.81</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-61.39</td>
<td>-19.82</td>
<td>-0.00</td>
<td>-0.03</td>
<td>0.01</td>
<td>141.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>21</td>
<td>872.77</td>
<td>2737.51</td>
<td>-0.01</td>
<td>1.76</td>
<td>-0.27</td>
<td>1193.12</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-872.77</td>
<td>2343.13</td>
<td>-0.01</td>
<td>1.76</td>
<td>-0.27</td>
<td>2343.13</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>21</td>
<td>-725.43</td>
<td>3432.33</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-10005.50</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>725.43</td>
<td>395.71</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>20</td>
<td>208.72</td>
<td>-1890.44</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>-8744.57</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-208.72</td>
<td>2879.93</td>
<td>-0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>2879.93</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>1659.58</td>
<td>5774.23</td>
<td>-0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>11266.44</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>-1659.58</td>
<td>-10957.92</td>
<td>-0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>10957.92</td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>61.39</td>
<td>-6.07</td>
<td>0.00</td>
<td>0.03</td>
<td>-0.01</td>
<td>-141.09</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-61.39</td>
<td>53.20</td>
<td>-0.00</td>
<td>-0.03</td>
<td>0.01</td>
<td>153.20</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>22</td>
<td>872.77</td>
<td>-450.00</td>
<td>-0.01</td>
<td>-1.64</td>
<td>-0.27</td>
<td>-975.62</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-872.77</td>
<td>-9929.10</td>
<td>-0.01</td>
<td>-1.64</td>
<td>-0.27</td>
<td>-9929.10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>22</td>
<td>-725.43</td>
<td>3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-395.71</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>725.43</td>
<td>9929.10</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>9929.10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>22</td>
<td>208.72</td>
<td>-1180.40</td>
<td>0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-2888.51</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-208.72</td>
<td>-2888.51</td>
<td>0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-2888.51</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>22</td>
<td>1659.58</td>
<td>5702.33</td>
<td>-0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>10538.13</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-1659.58</td>
<td>-3068.23</td>
<td>-0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-3068.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>61.39</td>
<td>-6.07</td>
<td>0.00</td>
<td>-0.03</td>
<td>0.01</td>
<td>141.09</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-61.39</td>
<td>-141.09</td>
<td>0.00</td>
<td>-0.03</td>
<td>0.01</td>
<td>141.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>872.77</td>
<td>-450.00</td>
<td>-0.01</td>
<td>1.64</td>
<td>-0.27</td>
<td>975.62</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-872.77</td>
<td>-975.62</td>
<td>0.01</td>
<td>1.64</td>
<td>-0.27</td>
<td>975.62</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>-725.43</td>
<td>3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-395.71</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>725.43</td>
<td>-395.71</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>10</td>
<td>208.72</td>
<td>-1180.40</td>
<td>0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-2888.51</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-208.72</td>
<td>-2888.51</td>
<td>0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-2888.51</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>10</td>
<td>1659.58</td>
<td>5702.33</td>
<td>-0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>10538.13</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-1659.58</td>
<td>-3068.23</td>
<td>-0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-3068.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>61.39</td>
<td>-6.07</td>
<td>0.00</td>
<td>-0.03</td>
<td>0.01</td>
<td>141.09</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-61.39</td>
<td>-141.09</td>
<td>0.00</td>
<td>-0.03</td>
<td>0.01</td>
<td>141.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>872.77</td>
<td>-450.00</td>
<td>-0.01</td>
<td>1.64</td>
<td>-0.27</td>
<td>975.62</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-872.77</td>
<td>-975.62</td>
<td>0.01</td>
<td>1.64</td>
<td>-0.27</td>
<td>975.62</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>-725.43</td>
<td>3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-395.71</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>725.43</td>
<td>-395.71</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>10</td>
<td>208.72</td>
<td>-1180.40</td>
<td>0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-2888.51</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-208.72</td>
<td>-2888.51</td>
<td>0.01</td>
<td>-1.67</td>
<td>-0.24</td>
<td>-2888.51</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

DYNAMIC ANALYSIS FOR SEISMIC LOADS

MEMBER END FORCES
STRUCTURE TYPE = SPACE

---

All units are - KGS METE (LOCAL)

------------------

STAAD.Pro 4622 User Manual
### Application Examples

#### EX. American Design Examples

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>4</td>
<td>118.30</td>
<td>-588.39</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.06</td>
<td>-323.84</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>4</td>
<td>-118.30</td>
<td>-588.39</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.06</td>
<td>323.84</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>-4.99</td>
<td>25.89</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.01</td>
<td>12.46</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4</td>
<td>4.99</td>
<td>25.89</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.01</td>
<td>-12.46</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>4</td>
<td>118.30</td>
<td>-588.39</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.06</td>
<td>-323.84</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4</td>
<td>-118.30</td>
<td>-588.39</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.06</td>
<td>323.84</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>6</td>
<td>25.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>-137.01</td>
<td>25.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.23</td>
<td>22.71</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4</td>
<td>137.01</td>
<td>25.89</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.23</td>
<td>-22.71</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>6</td>
<td>118.26</td>
<td>588.39</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.06</td>
<td>311.99</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4</td>
<td>-118.26</td>
<td>588.39</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.06</td>
<td>-311.99</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>1</td>
<td>8</td>
<td>5.97</td>
<td>25.89</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>4</td>
<td>-5.97</td>
<td>25.89</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>6</td>
<td>118.26</td>
<td>588.39</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.06</td>
<td>311.99</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>4</td>
<td>-118.26</td>
<td>588.39</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.06</td>
<td>-311.99</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>8</td>
<td>132.01</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.22</td>
<td>-10.26</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4</td>
<td>-132.01</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.22</td>
<td>10.26</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

**DYNAMIC ANALYSIS FOR SEISMIC LOADS**

**All units are** KGS METE (Local)

**Member End Forces** Structure Type = SPACE

---

**STAAD.Pro User Manual**
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>Load</th>
<th>Parameter Code</th>
<th>Critical Condition</th>
<th>Ratio</th>
<th>Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>15467.09</td>
<td>30.17</td>
<td>0.69</td>
<td>-0.10</td>
</tr>
<tr>
<td>28</td>
<td>185.42</td>
<td>0.01</td>
<td>-0.00</td>
<td>0.02</td>
</tr>
<tr>
<td></td>
<td>-104.72</td>
<td>25.75</td>
<td>-0.01</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>4933.53</td>
<td>21.16</td>
<td>0.65</td>
<td>-0.10</td>
</tr>
<tr>
<td>22</td>
<td>-4933.53</td>
<td>-21.16</td>
<td>-0.65</td>
<td>0.10</td>
</tr>
<tr>
<td>3</td>
<td>14950.07</td>
<td>-119.78</td>
<td>0.66</td>
<td>-0.10</td>
</tr>
<tr>
<td></td>
<td>-14950.07</td>
<td>119.78</td>
<td>-0.66</td>
<td>0.10</td>
</tr>
<tr>
<td>11</td>
<td>20238.35</td>
<td>175.70</td>
<td>0.66</td>
<td>-0.10</td>
</tr>
<tr>
<td></td>
<td>-20188.32</td>
<td>-175.70</td>
<td>-0.66</td>
<td>0.10</td>
</tr>
<tr>
<td>27</td>
<td>18</td>
<td>25.89</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td>0.02</td>
<td>1125.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>3</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td></td>
<td>0.00</td>
<td>1150.89</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>11</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>12</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
</tbody>
</table>

---

**DYNAMIC ANALYSIS FOR SEISMIC LOADS**

**--- PAGE NO. 20 ---**

**STAAD.PRO CODE CHECKING - S16-14 (v1.0)**

---

**ALL UNITS ARE - KN MET (UNLESS OTHERWISE NOTED)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Table</th>
<th>Result/</th>
<th>Critical Cond/</th>
<th>Ratio/</th>
<th>Loading/</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST</td>
<td>W310X97</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**END OF DESIGN OUTPUT OF MEMBER**

1. **ST**

2. **ST**

3. **ST**

4. **ST**

5. **ST**

6. **ST**
### Application Examples

**EX. American Design Examples**

<table>
<thead>
<tr>
<th>Member</th>
<th>Design Information</th>
<th>Result/</th>
<th>Critical Cond/</th>
<th>Ratio/</th>
<th>Loading/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>END OF DESIGN OUTPUT OF MEMBER</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7 ST</td>
<td>W310X97 (CANADIAN SECTIONS)</td>
<td>PASS Cl. 13.8.3</td>
<td>0.321</td>
<td>12</td>
<td>73.95 C</td>
</tr>
</tbody>
</table>

| 8 ST   | W310X97 (CANADIAN SECTIONS) | PASS Cl. 13.8.3 | 0.467 | 12 | 71.97 C | -1.22 | 169.22 | 1.80 |

**DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 21**

| 9 ST   | W310X97 (CANADIAN SECTIONS) | PASS Cl. 13.9.1 | 0.464 | 12 | 65.24 T | -1.21 | 168.84 | 0.00 |

| 10 ST  | W310X97 (CANADIAN SECTIONS) | PASS Cl. 13.8.3 | 0.324 | 11 | 73.92 C | 0.64 | -114.18 | 3.50 |

| 11 ST  | W310X97 (CANADIAN SECTIONS) | PASS Cl. 13.8.3 | 0.471 | 11 | 71.94 C | -1.22 | -171.07 | 1.80 |

| 12 ST  | W310X97 (CANADIAN SECTIONS) | PASS Cl. 13.9.1 | 0.467 | 11 | 59.54 T | -1.21 | -170.65 | 0.00 |

| 13 ST  | W250X39 (CANADIAN SECTIONS) | PASS Cl. 13.9.1 | 0.864 | 12 | 122.21 T | -0.02 | 106.25 | 1.80 |

| 14 ST  | W250X39 (CANADIAN SECTIONS) | PASS Cl. 13.8 | 0.798 | 12 | 16.27 C | 0.00 | 110.49 | 0.00 |

| 15 ST  | W250X39 (CANADIAN SECTIONS) | PASS Cl. 13.8 | 0.776 | 11 | 15.71 C | 0.00 | 107.46 | 3.00 |

| 16 ST  | W250X39 (CANADIAN SECTIONS) | PASS Cl. 13.9.1 | 0.844 | 11 | 123.57 T | -0.02 | 103.34 | 0.00 |

| 17 ST  | W250X39 (CANADIAN SECTIONS) | PASS Cl. 13.9.1 | 0.864 | 12 | 122.21 T | 0.02 | 106.25 | 1.80 |

**DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 22**

### STAAD.Pro Code Checking - S16-14 (v1.0)

**ALL UNITS ARE - KN MET (UNLESS OTHERWISE NOTED)**

---

**STAAD.Pro 4625 User Manual**
<table>
<thead>
<tr>
<th>Member</th>
<th>Section</th>
<th>Cl.</th>
<th>Factor</th>
<th>Design Moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST 19</td>
<td>W250X39</td>
<td>13.8</td>
<td>0.776</td>
<td>107.46 3.00</td>
</tr>
<tr>
<td>ST 20</td>
<td>C200X17</td>
<td>13.9.1</td>
<td>0.844</td>
<td>103.34 0.00</td>
</tr>
<tr>
<td>ST 21</td>
<td>C200X17</td>
<td>13.9.1</td>
<td>0.009</td>
<td>103.34 0.00</td>
</tr>
<tr>
<td>ST 22</td>
<td>C200X17</td>
<td>13.8.3</td>
<td>0.118</td>
<td>103.34 3.00</td>
</tr>
<tr>
<td>ST 23</td>
<td>L152X152X13</td>
<td>13.8.3</td>
<td>0.632</td>
<td>200.71 2.48</td>
</tr>
<tr>
<td>ST 24</td>
<td>L152X152X13</td>
<td>13.8.3</td>
<td>0.632</td>
<td>200.71 2.48</td>
</tr>
<tr>
<td>ST 25</td>
<td>C200X17</td>
<td>13.8.3</td>
<td>0.444</td>
<td>0.00 1.50</td>
</tr>
</tbody>
</table>

---

**Note:** OMEGA1 and OMEGA2 are calculated using C/S MOMENT OF ANALYTICAL MEMBERS.

---

**Dynamic Analysis for Seismic Loads**

---

**Finish**

---

**For technical assistance on STAAD.Pro, please visit**

**http://www.bentley.com/en/support/**

**Details about additional assistance from**

**Bentley and Partners can be found at program menu**

**Help->Technical Support**
EX. British Design Examples

EX. UK-1 Plane Frame with Steel Design

After one analysis, member selection is requested. Since member sizes change during the member selection, another analysis is done followed by final code checking to verify that the final sizes meet the requirements of the code based on the latest analysis results.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-1 Plane Frame with Steel Design.STD when you install the program.
where:

\[
L = 4.5 \, \text{m}, \quad H_1 = 6.0 \, \text{m}, \quad H_2 = 4.5 \, \text{m}, \quad \text{and} \quad H_3 = 2.7 \, \text{m}
\]
\[
WL_x = 9.0 \, \text{KN/m}, \quad WL_y = 15 \, \text{KN/m}, \quad RL = 13.5 \, \text{KN/m}, \quad LL = 17.5 \, \text{KN/m}, \quad P_1 = 135 \, \text{KN}, \quad P_2 = 35 \, \text{KN}
\]

Members 1, 3, & 4 are a UC356X368X129, Members 5, 6, & 7 are a UB533X210X82, Member 2 is a UC254X254X73, Members 8 through 13 are a UB457X152X52. All other members comprising the truss are a UA100X100X8

Actual input is shown in bold lettering followed by explanation.

**STAAD PLANE EXAMPLE PROBLEM NO. 1**

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

**UNIT METER KN**
Defines the input units for the data that follows.

**JOINT COORDINATES**

```
1 0 0 ; 2 9 0 ; 3 0 6 ; 4 3 6
5 6 6 ; 6 9 6 ; 7 0 10.5
8 9 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
14 3 12.3 ; 15 6 12.3 ; 16 4.5 13.2
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

**MEMBER INCIDENCE**

```
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
```

Defines the members by the joints to which they are connected.

**MEMBER PROPERTY BRITISH**

```
1 3 4 TA ST UC356X368X129 ; 2 TA ST UC254X254X73
5 6 7 TA ST UB533X210X82 ; 8 TO 13 TA ST UB457X152X52
14 TO 23 TA ST UA100X100X8
```

Member properties are from the British steel table. The term ST stands for standard single section.

**MEMB TRUSS**

```
14 TO 23
```

The above command defines that members 14 through 23 are of type truss. This means that these members can carry only axial tension/compression and no moments.

**MEMB RELEASE**

```
5 START MZ
```

Member 5 has local moment-z (MZ) released at the start joint. This means that the member cannot carry any moment-z (i.e., strong axis moment) at node 3.

**UNIT KNS MMS**

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

The DEFINE MATERIAL START initiates input for a material definition, which can later be assigned to all members. STAAD.Pro has several built-in material definitions, including the ISOTROPIC STEEL material used here. The length unit is changed to MMS (millimeters) to facilitate the input in familiar units.
MATERIAL STEEL ALL
UNIT METER

The CONSTANT command initiates input for material constants or material definition references. The BETA command specifies that members 3 and 4 are rotated by 90 degrees around their own longitudinal axis. See G. 4.3 Relationship Between Global and Local Coordinates (on page 2301) for the definition of the beta angle.

SUPPORT
1 FIXED ; 2 PINNED

A fixed support is located at joint 1 and a pinned support at joint 2.

PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTY LIST 1 2 5 8 14

The above PRINT commands are self-explanatory. The LIST option restricts the print output to the members listed.

LOADING 1 DEAD AND LIVE LOAD

Load case 1 is initiated followed by a title.

SELFWEIGHT Y -1.0

One of the components of load case 1 is the selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

JOINT LOAD
4 5 FY -65. ; 11 FY -155.

Load 1 contains joint loads also. FY indicates that the load is a force in the global Y direction.

MEMB LOAD
8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5

Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The term UNI stands for uniformly distributed load. Loads are applied on members 6, and 8 to 13.

CALCULATE RAYLEIGH FREQUENCY

The above command at the end of load case 1, is an instruction to perform a natural frequency calculation based on the Rayleigh method using the data in the above load case.

LOADING 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 9.0 ; 8 TO 10 UNI Y -15.0

Load case 2 is initiated and contains several member loads.

* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75

The above command identifies a combination load (case no. 3) with a title. The second line provides the load cases and their respective factors used for the load combination.

Note: Any line beginning with an asterisk (* character) is treated as a comment line.

PERFORM ANALYSIS

This command instructs the program to proceed with the analysis.

LOAD LIST 1 3
The above command activates load cases 1 and 3 only for the commands to follow. This also means that load case 2 will be made inactive.

```
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
```

The above PRINT commands are self-explanatory. Also note that all the forces and reactions will be printed for load cases 1 and 3 only.

```
PARAMETER
CODE EN 1993-1-1:2005
NA 1
NSF 0.85 ALL
BEAM 1 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
```

The PARAMETER command is used to specify steel design parameters such as net section factor (NSF), effective length factor for bending about the minor axis (KY), etc. Information on these parameters can be obtained from the manual where the implementation of the code is explained. The BEAM parameter is specified to perform design at every 1/12th point along the member length, which is also the default. The RATIO parameter specifies that the ratio of actual loading over section capacity should not exceed 0.9.

```
SELECT ALL
```

The above command instructs the program to select the most economic section for all the members based on the results of the analysis.

```
GROUP MEMB 1 3 4
GROUP MEMB 5 6 7
GROUP MEMB 8 TO 13
GROUP MEMB 14 TO 23
```

Although the program selects the most economical section for all members, it is not always practical to use many different sizes in one structure. Grouping is a procedure by which the cross section which has the largest value for the specified attribute, which in this case is the default and hence the AREA, from among the associated member list, is assigned to all members in the list. Hence, the cross sections for members 1, 3 and 4 are replaced with the one with the largest area from among the three.

```
PERFORM ANALYSIS
```

As a result of the selection and grouping, the member sizes are no longer the same as the ones used in the original analysis. Hence, it is necessary to reanalyze the structure using the new properties to get new values of forces in the members.

```
PARAMETER
BEAM 1.0 ALL
RATIO 1.0 ALL
TRACK 1.0 ALL
```

A new set of values are now provided for the above parameters. The actual load to member capacity RATIO has been redefined as 1.0. The TRACK parameter tells the program to print out the design results to the intermediate level of descriptively.

```
CHECK CODE ALL
```

With the above command, the latest member sizes with the latest analysis results are checked to verify that they satisfy the CODE specifications.

```
STEEL TAKE OFF
```
This command instructs the program to list the length and weight of all the different member sizes.

**FINISH**

This command terminates the STAAD run.

**Input File**

STAAD PLANE EXAMPLE PROBLEM NO. 1
UNIT METER KN

JOINT COORDINATES
1 0. 0. ; 2 9. 0. ; 3 0. 6. ; 4 3. 6. 
5 6. 6. ; 6 9. 6. ; 7 0 10.5
8 9. 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
14 3. 12.3 ; 15 6. 12.3 ; 16 4.5 13.2

MEMBER INCIDENCE
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10

MEMBER PROPERTY BRITISH
1 3 4 TA ST UC356X368X129 ; 2 TA ST UC254X254X73
5 6 7 TA ST UB533X210X82 ; 8 TO 13 TA ST UB457X152X52
14 TO 23 TA RA UA100X100X8

*MEMB TRUSS
*14 TO 23

MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99

UNIT KNS MMS

DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL

CONSTANTS
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4

UNIT METER

SUPPORT
1 FIXED ; 2 PINNED

PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTY LIST 1 2 5 8 14
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0

JOINT LOAD
4 5 FY -65. ; 11 FY -155.

MEMB LOAD
8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
CALCULATE RAYLEIGH FREQUENCY
LOADING 2  WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 9.0 ; 8 TO 10 UNI Y -15.0
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
PARAMETER
CODE EN 1993-1-1:2005
NA 1
NSF 0.85 ALL
BEAM 3 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
SELECT ALL
GROUP MEMB 1 3 4
GROUP MEMB 5 6 7
GROUP MEMB 8 TO 13
GROUP MEMB 14 TO 23
PERFORM ANALYSIS
PARAMETER
BEAM 3 ALL
RATIO 1.0 ALL
TRACK 1.0 ALL
CHECK CODE ALL
STEEL TAKE OFF
FINISH

STAAD Output File

*********************
*  STAAD.Pro CONNECT Edition  *
*  Version 22.01.00.**      *
*  Proprietary Program of   *
*  Bentley Systems, Inc.    *
*  Date= APR 14, 2019       *
*  Time= 22:52:36           *
*  *  Licensed to: Bentley Systems Inc  *
*********************

1. STAAD PLANE EXAMPLE PROBLEM NO. 1
INPUT FILE: UK-1 Plane Frame with Steel Design.STD
2. UNIT METER KN
3. JOINT COORDINATES
4. 1 0. 0. ; 2 9. 0. ; 3 0. 6. ; 4 3. 6.
5. 5 6. 6. ; 6 9. 6. ; 7 0 10.5
6. 8 9. 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
7. 11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
8. 14 3. 12.3 ; 15 6. 12.3 ; 16 4.5 13.2
9. MEMBER INCIDENCE
10. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
11. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
12. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
13. 1 4 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
14. 18 10 15 ; 19 10 13 ; 20 7 9
15. 21 9 11 ; 22 10 11 ; 23 8 10
16. MEMBER PROPERTY BRITISH
17. 1 3 4 TA ST UC356X368X129 ; 2 TA ST UC254X254X73
18. 5 6 7 TA ST UB533X210X82 ; 8 TO 13 TA ST UB457X152X52
19. 14 TO 23 TA RA UA100X100X8
20. *MEMB TRUSS
21. *14 TO 23
22. MEMB RELEASE
23. 5 START MZ
24. 14 TO 23 START MPY 0.99 MPZ 0.99
25. 14 TO 23 END MPY 0.99 MPZ 0.99
26. UNIT KNS MMS
27. DEFINE MATERIAL START
28. ISOTROPIC STEEL
29. E 210
30. POISSON 0.3
31. DENSITY 7.68191E-008
32. ALPH 6E-006
33. DAMP 0.03
34. TYPE STEEL
35. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
36. END DEFINE MATERIAL
37. CONSTANTS
38. MATERIAL STEEL ALL

EXAMPLE PROBLEM NO. 1

-- PAGE NO. 2
39. BETA 90.0 MEMB 3 4
40. UNIT METER
41. SUPPORT
42. 1 FIXED ; 2 PINNED
43. PRINT MEMBER INFORMATION LIST 1 5 14

MEMBER INFORMATION

------------------
MEMBER    START    END         LENGTH     BETA
JOINT   JOINT        (METE)     (DEG)     RELEASES
1        1       3           6.000     0.00    000000000000
5        3       4           3.000     0.00    000000000000
14        9      12           1.172     0.00    000000000000
************ END OF DATA FROM INTERNAL STORAGE ************
44. PRINT MEMBER PROPERTY LIST 1 2 5 8 14

MEMBER PROPERTY LIST

---------------
MEMBER   PROFILE              AX/          IZ/          IY/        IX/
AY           AZ           SZ         SY
1  ST  UC356X368X129     164.00     40200.00     14600.00      152.61
   36.98       116.11      2260.97      792.19
2  ST  UC254X254X73      93.10      11400.00      3910.00       57.62
   21.85        65.07       897.28       307.15
5  ST  UB533X210X82     105.00      47500.00      2010.00       51.52
   50.72       49.61      1798.22      192.53
8  ST  UB457X152X52      66.60      21400.00       645.00      21.37
   34.18        29.90       951.53       84.65

STAAD.Pro
4634
User Manual
APPLICATION EXAMPLES

EX. British Design Examples

14 RA UA100X100X8  15.50  236.82  59.54  3.31
  5.33  5.33  33.49  15.20
************** END OF DATA FROM INTERNAL STORAGE **************
45. LOADING 1 DEAD AND LIVE LOAD
46. SELFWEIGHT Y -1.0
47. JOINT LOAD
48. 4 S FY -65.; 11 FY -155.
49. MEMB LOAD
50. 8 TO 13 UNI Y -13.5; 6 UNI GY -17.5
51. CALCULATE RAYLEIGH FREQUENCY
52. LOADING 2 WIND FROM LEFT
53. MEMBER LOAD
54. 1 2 UNI GX 9.0; 8 TO 10 UNI Y -15.0
55. * 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
56. LOAD COMB 3 75 PERCENT DL LL WL
57. 1 0.75 2 0.75
58. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS        16  NUMBER OF MEMBERS      23
NUMBER OF PLATES         0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES       0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
EXAMPLE PROBLEM NO. 1 -- PAGE NO. 5
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      43
TOTAL LOAD COMBINATION CASES =     1  SO FAR.
**********************************************************
*                                                        *
* RAYLEIGH FREQUENCY FOR LOADING 1 =    3.48541 CPS  *
* MAX DEFLECTION =  2.54900 CM  GLO X, AT JOINT 7  *
*                                                        *
**********************************************************
59. LOAD LIST 1 3
60. PRINT MEMBER FORCES
MEMBER FORCES
EXAMPLE PROBLEM NO. 1 -- PAGE NO. 6
MEMBER END FORCES  STRUCTURE TYPE = PLANE
-----------------------------------
ALL UNITS ARE -- KNS  METE  (LOCAL )
MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
1   1   1  239.70 -7.78  0.00  0.00  0.00  -72.75
  3 -232.14  7.78  0.00  0.00  0.00  26.08
  3  179.63  85.56  0.00  0.00  0.00  334.46
  3 -173.96 -45.06  0.00  0.00  0.00  57.40
2   1   3  150.33 -22.49  0.00  0.00  0.00  -26.08
  7  147.12  22.49  0.00  0.00  0.00  -75.14
  3  128.83  0.18  0.00  0.00  0.00  -57.40
  7 -126.42 30.56  0.00  0.00  0.00  -11.77
  3  1   2  258.31  0.00 -7.78  0.00  0.00  0.00
  6 -250.75  0.00  7.78  0.00 46.67  0.00
  3  2  244.50  0.00 -15.69  0.00  0.00  0.00
  6 -238.83  0.00  15.69  0.00 94.14  0.00
4   1   6  142.80  0.00 -22.49  0.00 71.03  0.00
  8 -137.13  0.00  22.49  0.00 38.19  0.00
  3  6  141.64  0.00  60.93  0.00 140.17  0.00
  8 -137.39  0.00  60.93  0.00 134.03  0.00
5   1   3 -14.72  81.80  0.00  0.00  0.00  0.00

STAAD.Pro 4635 User Manual
**Application Examples**

**EX. British Design Examples**

---

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>12</td>
<td>167.89</td>
<td>34.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-31.32</td>
<td></td>
</tr>
<tr>
<td></td>
<td>14</td>
<td>-167.55</td>
<td>3.06</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>59.17</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>14</td>
<td>188.07</td>
<td>-88.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-75.98</td>
<td></td>
</tr>
<tr>
<td></td>
<td>16</td>
<td>-187.61</td>
<td>112.53</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-99.54</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>15</td>
<td>188.94</td>
<td>-88.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-76.86</td>
<td></td>
</tr>
<tr>
<td></td>
<td>16</td>
<td>-187.58</td>
<td>112.58</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-99.54</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>13</td>
<td>182.56</td>
<td>36.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-33.87</td>
<td></td>
</tr>
<tr>
<td></td>
<td>15</td>
<td>-182.10</td>
<td>-11.92</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>76.05</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>8</td>
<td>184.91</td>
<td>48.81</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>30.19</td>
<td></td>
</tr>
<tr>
<td></td>
<td>13</td>
<td>-184.45</td>
<td>-24.43</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>33.87</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>9</td>
<td>-6.25</td>
<td>3.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.08</td>
<td></td>
</tr>
<tr>
<td></td>
<td>12</td>
<td>6.36</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.08</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>9</td>
<td>4.86</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.08</td>
<td></td>
</tr>
<tr>
<td></td>
<td>14</td>
<td>-4.64</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>-24.97</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.01</td>
<td></td>
</tr>
<tr>
<td></td>
<td>14</td>
<td>25.13</td>
<td>0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>11</td>
<td>-107.59</td>
<td>0.08</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.01</td>
<td></td>
</tr>
<tr>
<td></td>
<td>14</td>
<td>107.80</td>
<td>0.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.01</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>11</td>
<td>-107.68</td>
<td>0.06</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.01</td>
<td></td>
</tr>
<tr>
<td></td>
<td>15</td>
<td>107.84</td>
<td>0.07</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>10</td>
<td>-10.44</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.01</td>
<td></td>
</tr>
<tr>
<td></td>
<td>15</td>
<td>10.66</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.01</td>
<td></td>
</tr>
</tbody>
</table>
### Example Problem No. 1

**Structure Type:** Plane

**Units:** KNS, METE (Local)

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>Joint</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Moment-Y</th>
<th>Moment-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>10</td>
<td>26.57</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>15</td>
<td>-26.41</td>
<td>0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>12.13</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>13</td>
<td>-12.02</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>-32.23</td>
<td>0.04</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>13</td>
<td>32.31</td>
<td>0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>-85.02</td>
<td>0.14</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>85.02</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>-95.19</td>
<td>0.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>95.19</td>
<td>0.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>-90.88</td>
<td>0.14</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>90.88</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>-66.58</td>
<td>0.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>66.58</td>
<td>0.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>22</td>
<td>1</td>
<td>-99.16</td>
<td>0.14</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>99.16</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>-25.81</td>
<td>0.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>25.81</td>
<td>0.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>-110.95</td>
<td>0.14</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>110.95</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>8</td>
<td>5.05</td>
<td>0.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-5.05</td>
<td>0.09</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

### Support Reaction

**Structure Type:** Plane

**Units:** KNS, METE

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>7.78</td>
<td>239.70</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-72.75</td>
</tr>
<tr>
<td>3</td>
<td>-85.56</td>
<td>179.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>334.46</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-7.78</td>
<td>258.31</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>15.69</td>
<td>244.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

### Parameter

61. Print Support Reaction

62. Parameter

63. Code EN 1993-1-1:2005

64. NA 1

65. NSF 0.85 ALL

66. Beam 3 ALL

67. KY 1.2 MEMB 3 4

68. Ratio 0.9 ALL

69. Select All

### Steel Design

**STAAD.Pro Member Selection - BS EN 1993-1-1:2005**

**National Annex - NA to BS EN 1993-1-1:2005**

**Program Code Revision V1.13 BS_EC3_2005/1**

**Example Problem No. 1**

All units are KNS, METE (unless otherwise noted)

<table>
<thead>
<tr>
<th>Member</th>
<th>Table</th>
<th>Result/ Critical Cond/ Ratio/ Loading/</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST</td>
<td>UC305X305X118 (British Sections)</td>
<td>------------------</td>
</tr>
</tbody>
</table>
## Application Examples
### EX. British Design Examples

<table>
<thead>
<tr>
<th>Member</th>
<th>Table</th>
<th>Result/</th>
<th>Critical Cond/</th>
<th>Ratio/</th>
<th>Loading/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
</tr>
<tr>
<td>11 ST</td>
<td>UB406X140X39 (BRITISH SECTIONS)</td>
<td>PASS</td>
<td>EC-6.3.3-662</td>
<td>0.803</td>
<td></td>
</tr>
<tr>
<td>12 ST</td>
<td>UB254X146X37 (BRITISH SECTIONS)</td>
<td>PASS</td>
<td>EC-6.3.3-662</td>
<td>0.870</td>
<td></td>
</tr>
<tr>
<td>13 ST</td>
<td>UB356X171X45 (BRITISH SECTIONS)</td>
<td>PASS</td>
<td>EC-6.3.3-662</td>
<td>0.851</td>
<td></td>
</tr>
<tr>
<td>14 RA</td>
<td>UA50x50x4 (BRITISH SECTIONS)</td>
<td>PASS</td>
<td>EC-6.3.3-662</td>
<td>0.746</td>
<td></td>
</tr>
</tbody>
</table>
### Example Problem No. 1

---

**All Units Are - KNS METE (Unless Otherwise Noted)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Table</th>
<th>Result/</th>
<th>Critical Cond/</th>
<th>Ratio/</th>
<th>Loading/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Location</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Location</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>RA UA65x50x5 (BRITISH SECTIONS)</td>
<td>PASS</td>
<td>EC-6.2.9.2/3</td>
<td>0.826</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>RA UA60x40x6 (BRITISH SECTIONS)</td>
<td>PASS</td>
<td>EC-6.2.9.2/3</td>
<td>0.888</td>
<td></td>
</tr>
</tbody>
</table>

---

**Example Problem No. 1**

---

**Grouping Based on Member**

- **Group Mem 1 3 4**: Based on **Member 1** (ST UC305X305X118) List= 1....
- **Group Mem 5 6 7**: Based on **Member 7** (ST UB457X191X67) List= 5....
- **Group Mem 8 to 13**: Based on **Member 13** (ST UB356X171X45) List= 8....
- **Group Mem 14 to 23**: Based on **Member 18** (RA UA60x60x5) List= 14....

**Perform Analysis**

---

**Rayleigh Frequency for Loading**: 1 = 2.68348 CPS

**Max Deflection**: 4.57423 CM GLO X, AT JOINT 7
**Application Examples**

**EX. British Design Examples**

```plaintext
75. PARAMETER
76. BEAM 3 ALL
77. RATIO 1.0 ALL
--- PAGE NO. 13
78. TRACK 1.0 ALL
79. CHECK CODE ALL

STEEL DESIGN

STAAD.PRO CODE CHECKING - BS EN 1993-1-1:2005

NATIONAL ANNEX - NA to BS EN 1993-1-1:2005

PROGRAM CODE REVISION V1.13 BS_EC3_2005/1

--- PAGE NO. 14

ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/ CRITICAL COND/ RATIO/ LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
</tr>
</tbody>
</table>

|=======================================================================|
| 1 ST UC305X305X118(BRITISH SECTIONS) | PASS | EC-6.3.3-662 | 0.910 |
| 3 | 178.79 C | 0.00 | 351.35 | 0.00 |
| --------------------------------- | ------ | ------------ | -------- |
| CALCULATED CAPACITIES FOR MEMB 1 UNIT - kN,m SECTION CLASS 1 | MCZ= 460.6 MCY= 210.3 PC= 2285.0 PT= 3525.0 MB= 410.5 PV= 582.9 | BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000 | PZ= 3525.00 FX/PZ = 0.05 MRZ= 460.6 MRY= 210.3 |
| 2 ST UC303X203X46 (BRITISH SECTIONS) | PASS | EC-6.3.3-662 | 0.802 |
| 1 | 144.94 C | 0.00 | -68.59 | 4.50 |
| --------------------------------- | ------ | ------------ | -------- |
| CALCULATED CAPACITIES FOR MEMB 2 UNIT - kN,m SECTION CLASS 1 | MCZ= 116.8 MCY= 54.3 PC= 800.0 PT= 1379.5 MB= 98.9 PV= 229.9 | BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000 | PZ= 1379.45 FX/PZ = 0.11 MRZ= 116.8 MRY= 54.3 |
| 3 ST UC305X305X118(BRITISH SECTIONS) | PASS | EC-6.2.9.1 | 0.438 |
| 3 | 235.25 C | 92.19 | 0.00 | 6.00 |
| --------------------------------- | ------ | ------------ | -------- |
| CALCULATED CAPACITIES FOR MEMB 3 UNIT - kN,m SECTION CLASS 1 | MCZ= 460.6 MCY= 210.3 PC= 1930.8 PT= 3525.0 MB= 410.5 PV= 582.9 | BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000 | PZ= 3525.00 FX/PZ = 0.07 MRZ= 460.6 MRY= 210.3 |
| 4 ST UC305X305X118(BRITISH SECTIONS) | PASS | EC-6.2.9.1 | 0.659 |
| 3 | 138.96 C | 138.54 | 0.00 | 0.00 |
| --------------------------------- | ------ | ------------ | -------- |
| CALCULATED CAPACITIES FOR MEMB 4 UNIT - kN,m SECTION CLASS 1 | MCZ= 460.6 MCY= 210.3 PC= 2466.9 PT= 3525.0 MB= 438.0 PV= 582.9 | BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000 | PZ= 3525.00 FX/PZ = 0.04 MRZ= 460.6 MRY= 210.3 |
```
MEMBER | TABLE | RESULT/CRITICAL COND/ RATIO/ LOADING/
FX | MY | MZ | LOCATION
---|---|---|---
5 ST UB457X191X67 (BRITISH SECTIONS)
PASS | EC-6.3.2 LTB | 0.861
| 14.61 T | 0.00 | 241.23 | 3.00

CALCULATED CAPACITIES FOR MEMB 5 UNIT - kN,m SECTION CLASS 1
MCZ= 345.5 MCY= 55.7 PC= 1485.4 PT= 2009.3 MB= 280.1 PV= 555.4
BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000
PZ= 2009.25 FX/PZ = 0.01 MRZ= 345.5 MRY= 55.7

6 ST UB457X191X67 (BRITISH SECTIONS)
PASS | EC-6.3.2 LTB | 0.882
| 14.61 T | 0.00 | -246.94 | 0.75

CALCULATED CAPACITIES FOR MEMB 6 UNIT - kN,m SECTION CLASS 1
MCZ= 345.5 MCY= 55.7 PC= 1485.4 PT= 2009.3 MB= 280.1 PV= 555.4
BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000
PZ= 2009.25 FX/PZ = 0.01 MRZ= 345.5 MRY= 55.7

7 ST UB457X191X67 (BRITISH SECTIONS)
PASS | EC-6.3.2 LTB | 0.824
| 44.31 T | 0.00 | -230.73 | 3.00

CALCULATED CAPACITIES FOR MEMB 7 UNIT - kN,m SECTION CLASS 1
MCZ= 345.5 MCY= 55.7 PC= 1485.4 PT= 2009.3 MB= 280.1 PV= 555.4
BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000
PZ= 2009.25 FX/PZ = 0.02 MRZ= 345.5 MRY= 55.7

8 ST UB356X171X45 (BRITISH SECTIONS)
PASS | EC-6.3.3-662 | 0.512
| 165.72 C | 0.00 | 68.59 | 0.00

CALCULATED CAPACITIES FOR MEMB 8 UNIT - kN,m SECTION CLASS 1
MCZ= 182.1 MCY= 34.5 PC= 1193.5 PT= 1346.6 MB= 175.5 PV= 363.1
BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000
PZ= 1346.55 FX/PZ = 0.12 MRZ= 182.1 MRY= 34.5

9 ST UB356X171X45 (BRITISH SECTIONS)
PASS | EC-6.3.3-662 | 0.573
| 166.04 C | 0.00 | 77.15 | 1.75

CALCULATED CAPACITIES FOR MEMB 9 UNIT - kN,m SECTION CLASS 1
MCZ= 182.1 MCY= 34.5 PC= 1193.5 PT= 1346.6 MB= 175.5 PV= 363.1
BUCKLING CO-EFFICIENTS C1 AND K : C1 = 1.127 K = 1.000

---

**Application Examples**

**EX. British Design Examples**

---

---
### Example Problem No. 1

**-- Page No. 17**

**All Units Are - KNs Mete (Unless Otherwise Noted)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Table</th>
<th>Result</th>
<th>Critical Cond</th>
<th>Ratio</th>
<th>Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td></td>
<td>Location</td>
</tr>
</tbody>
</table>

---

**Example 1**

<table>
<thead>
<tr>
<th>PZ= 1346.55</th>
<th>FX/PZ = 0.12</th>
<th>MRZ= 182.1</th>
<th>MRY= 34.5</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>10 ST UB356X171X45 (BRITISH SECTIONS)</th>
<th>PASS</th>
<th>EC-6.3.3-662</th>
<th>0.682</th>
</tr>
</thead>
<tbody>
<tr>
<td>186.60 C</td>
<td>0.00</td>
<td>-97.18</td>
<td>1.75</td>
</tr>
</tbody>
</table>

### Calculated Capacities for Memb 10 Unit - kN.m Section Class 1

- $MCZ = 182.1$ (kN.m)
- $MCY = 34.5$ (kN.m)
- $PC = 1193.5$ (kN.m)
- $PT = 1346.6$ (kN.m)
- $MB = 175.5$ (kN.m)
- $PV = 363.1$ (kN.m)

**Buckling Co-Efficients $C1$ and $K$:**

- $C1 = 1.127$
- $K = 1.000$

<table>
<thead>
<tr>
<th>PZ= 1346.55</th>
<th>FX/PZ = 0.14</th>
<th>MRZ= 182.1</th>
<th>MRY= 34.5</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>11 ST UB356X171X45 (BRITISH SECTIONS)</th>
<th>PASS</th>
<th>EC-6.3.3-662</th>
<th>0.682</th>
</tr>
</thead>
<tbody>
<tr>
<td>186.46 C</td>
<td>0.00</td>
<td>-97.18</td>
<td>1.75</td>
</tr>
</tbody>
</table>

### Calculated Capacities for Memb 11 Unit - kN.m Section Class 1

<table>
<thead>
<tr>
<th>PZ= 1346.55</th>
<th>FX/PZ = 0.14</th>
<th>MRZ= 182.1</th>
<th>MRY= 34.5</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>12 ST UB356X171X45 (BRITISH SECTIONS)</th>
<th>PASS</th>
<th>EC-6.3.3-662</th>
<th>0.584</th>
</tr>
</thead>
<tbody>
<tr>
<td>175.61 C</td>
<td>0.00</td>
<td>77.56</td>
<td>1.75</td>
</tr>
</tbody>
</table>

### Calculated Capacities for Memb 12 Unit - kN.m Section Class 1

<table>
<thead>
<tr>
<th>PZ= 1346.55</th>
<th>FX/PZ = 0.13</th>
<th>MRZ= 182.1</th>
<th>MRY= 34.5</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>13 ST UB356X171X45 (BRITISH SECTIONS)</th>
<th>PASS</th>
<th>EC-6.3.3-662</th>
<th>0.831</th>
</tr>
</thead>
<tbody>
<tr>
<td>121.12 C</td>
<td>0.00</td>
<td>130.01</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Calculated Capacities for Memb 13 Unit - kN.m Section Class 1

<table>
<thead>
<tr>
<th>PZ= 1346.55</th>
<th>FX/PZ = 0.09</th>
<th>MRZ= 182.1</th>
<th>MRY= 34.5</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>14 RA UA60x60x5 (BRITISH SECTIONS)</th>
<th>PASS</th>
<th>EC-6.3.3-662</th>
<th>0.266</th>
</tr>
</thead>
<tbody>
<tr>
<td>19.99 C</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Calculated Capacities for Memb 14 Unit - kN.m Section Class 4

<table>
<thead>
<tr>
<th>PZ= 136.77</th>
<th>FX/PZ = 0.15</th>
<th>MRZ= 182.1</th>
<th>MRY= 34.5</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>15 RA UA60x60x5 (BRITISH SECTIONS)</th>
<th>PASS</th>
<th>EC-6.3.3-662</th>
</tr>
</thead>
<tbody>
<tr>
<td>136.77 C</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro**

4642

**User Manual**
**Application Examples**

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/CRITICAL COND/LOADING/RATIO/LOCATION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FX</td>
<td>MY</td>
</tr>
<tr>
<td></td>
<td>MCZ= 1.5 MCY= 0.8 PC= 35.3 PT= 136.8 MB= 1.6 PV= 27.1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PZ= 136.77 FX/PZ = 0.04 MRZ= 1.5 MRY= 0.8</td>
<td></td>
</tr>
</tbody>
</table>

**Example Problem No. 1**

---

**PASS** EC-6.3.3-662  0.168

<table>
<thead>
<tr>
<th>1</th>
<th>PASS</th>
<th>EC-6.2.9.2/3</th>
<th>0.797</th>
</tr>
</thead>
<tbody>
<tr>
<td>16 RA UA60x60x5 (BRITISH SECTIONS)</td>
<td>105.96 T</td>
<td>0.00</td>
<td>-0.02</td>
</tr>
<tr>
<td>17 RA UA60x60x5 (BRITISH SECTIONS)</td>
<td>104.19 T</td>
<td>0.00</td>
<td>-0.01</td>
</tr>
<tr>
<td>18 RA UA60x60x5 (BRITISH SECTIONS)</td>
<td>22.72 C</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>19 RA UA60x60x5 (BRITISH SECTIONS)</td>
<td>27.43 T</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>20 RA UA60x60x5 (BRITISH SECTIONS)</td>
<td>83.85 T</td>
<td>0.00</td>
<td>-0.03</td>
</tr>
</tbody>
</table>
### Calculated Capacities for Members

<table>
<thead>
<tr>
<th>Member</th>
<th>Table</th>
<th>Result/Location</th>
<th>Critical Condition</th>
<th>Ratio</th>
<th>Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td>21 RA</td>
<td>UA60x60x5</td>
<td>(BRITISH SECTIONS)</td>
<td>Pass EC-6.2.9.2/3</td>
<td>0.691</td>
<td></td>
</tr>
<tr>
<td>22 RA</td>
<td>UA60x60x5</td>
<td>(BRITISH SECTIONS)</td>
<td>Pass EC-6.2.9.2/3</td>
<td>0.738</td>
<td></td>
</tr>
<tr>
<td>23 RA</td>
<td>UA60x60x5</td>
<td>(BRITISH SECTIONS)</td>
<td>Pass EC-6.2.9.2/3</td>
<td>0.774</td>
<td></td>
</tr>
</tbody>
</table>

### Steel Take Off

<table>
<thead>
<tr>
<th>Profile</th>
<th>Length (Meters)</th>
<th>Weight (Kns)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ST UC305X305X118</td>
<td>16.50</td>
<td>19.013</td>
</tr>
<tr>
<td>ST UC283X283X46</td>
<td>4.50</td>
<td>2.029</td>
</tr>
<tr>
<td>ST UB457X191X67</td>
<td>9.00</td>
<td>5.911</td>
</tr>
<tr>
<td>ST UB356X171X45</td>
<td>10.50</td>
<td>4.620</td>
</tr>
<tr>
<td>RA UA60x60x5</td>
<td>19.93</td>
<td>0.891</td>
</tr>
</tbody>
</table>

TOTAL = 32.464

### Application Examples

**EX. British Design Examples**

---

**Application Examples**

**EX. British Design Examples**
EX. UK-2 Area Load Generation on Floor Structure

A floor structure (bound by global X-Z axis) made up of steel beams is subjected to area load (i.e., load/area of floor). Load generation based on one-way distribution is illustrated in this example.

In the case of loads such as joint loads and member loads, the magnitude and direction of the load at the applicable joints and members is directly known from the input. However, the area load is a different sort of load where a load intensity on the given area has to be converted to joint and member loads. The calculations required to perform this conversion are done only during the analysis. Consequently, the loads generated from the AREA LOAD command can be viewed only after the analysis is completed.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-2 Area Load Generation on Floor Structure.STD when you install the program.
Where:

\[ W1 = 1.5 \text{ m}, W2 = 1.0 \text{ m}, W3 = 1.5 \text{ m}, L1 = 2.0 \text{ m}, L2 = 4.5 \text{ m}, L3 = 3.0 \text{ m} \]

Actual input is shown in bold lettering followed by explanation.

**STAAD FLOOR A FLOOR FRAME DESIGN WITH AREA LOAD**

Every input has to start with the term STAAD. The term FLOOR signifies that the structure is a floor structure and the structure is in the \( x-z \) plane.

**UNIT METER KNS**

Defines the input units for the data that follows.

**JOINT COORDINATES**

\[
\begin{align*}
1 & 0 & 0 & 0 & 5 & 6 & 0 & 0 & ; & 7 & 1.5 & 0 & 3 \\
8 & 3 & 0 & 3 & ; & 9 & 4 & 0 & 3 & ; & 10 & 4.5 & 0 & 3 & ; & 11 & 5 & 0 & 3 \\
12 & 6 & 0 & 3 & ; & 13 & 0 & 0 & 7.5 & ; & 14 & 1.5 & 0 & . & 7.5 \\
15 & 3.5 & 0 & 7.5
\end{align*}
\]
Joint numbers followed by X, Y and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). Joints between 1 and 5 (i.e., 2, 3, 4) are generated in the first line of input, taking advantage of the equal spacing between the joints (see TR.11 Joint Coordinates Specification (on page 2425) for more information).

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCES
1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
19 7 14 ; 20 2 7 ; 21 9 15
22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
26 17 19 ; 27 12 17 ; 28 5 12

Defines the members by the joints to which they are connected.

MEMB PROP BRITISH
1 TO 28 TABLE ST UB305X165X40

Member properties are specified from the British steel table. The term ST stands for standard single section.

* MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
MEMB RELEASE
1 5 10 14 15 18 26 20 TO 24 START MZ
4 9 13 14 15 18 27 25 19 21 TO 24 END MZ

The first set of members (1 5 10 etc) have local moment-z (MZ) released at the start joint. This means that these members cannot carry any moment-z (i.e., strong axis moment) at the start joint. The second set of members have MZ released at the end joints.

Note: Any line beginning with an asterisk (*) character is treated as a comment line.

UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT METER

Define the material properties for steel. The units are changed to millimeters and then back to meters in order to facilitate inputting the values in familiar units.

CONSTANT
MATERIAL STEEL ALL

The CONSTANT command instructs the program to use the defined steel material for all members.

SUPPORT
1 5 13 17 20 21 FIXED
A fixed support has been specified at the above joints.

**LOADING 1 14.5 KN/sq.m. DL+LL**

Load case 1 is initiated followed by a title.

**FLOOR LOAD**

**YRANGE -0.5 0.5 FLOAD -14.5**

All members within a range of -0.5 meters to 0.5 meters in the global Y direction (which is the entire floor of this model) are subjected to an floor load of 14.5 KN/sq.m. The program converts area loads into individual member loads.

**PERFORM ANALYSIS PRINT LOAD DATA**

This command instructs the program to proceed with the analysis. The PRINT LOAD DATA command is specified to obtain a listing of the member loads which were generated from the FLOOR LOAD.

**PARAMETERS**

**CODE BRITISH**

**BEAM 1 ALL**

**DMAX 0.6 ALL**

**DMIN 0.3 ALL**

**UNL 0.3 ALL**

The PARAMETER command is used to specify steel design parameters (See Table 1 in D3.B.6 Design Parameters on page 1661). Design is to be performed per the specifications of the BS 5950 2000 Code. The BEAM parameter is specified to perform design at every 1/12th point along the member length. DMAX and DMIN specify maximum and minimum depth limitations to be used during member selection. UNL is used to specify unsupported length of the compression flange to be used for calculation of allowable bending stress.

**SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27**

The above command instructs the program to select the most economical section from the British steel table for the members listed.

**FINISH**

The FINISH command terminates the STAAD run.

**Input File**

**STAAD FLOOR A FLOOR FRAME DESIGN WITH AREA LOAD**

**UNIT METER KNS**

**JOINT COORDINATES**

1 0.0 0.5 6.0 0.0 ; 7 1.5 0.3
8 3.0 0.3 ; 9 4.0 0.3 ; 10 4.5 0.3
11 5.0 0.3 ; 12 6.0 0.3 ; 13 0.0 7.5
14 1.5 0 7.5 ; 15 3.5 0.0 7.5
16 5.0 7.5 ; 17 6.0 7.5 18 0.0 8.5
19 6.0 8.5 ; 20 0.0 10.5 ; 21 6.0 10.5

**MEMBER INCIDENCES**

1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
19 7 14 ; 20 2 7 ; 21 9 15
22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
26 17 19 ; 27 12 17 ; 28 5 12

**MEMB PROP BRITISH**

1 TO 28 TABLE ST UB305X165X40

* MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ

**MEMB RELEASE**
1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL ALL
UNIT METER KN
SUPPORT
1 5 13 17 20 21 FIXED
LOADING 1 14.5 KN/sq.m. DL+LL
FLOOR LOAD
YRANGE -0.5 0.5 FLOAD -14.5
PERFORM ANALYSIS PRINT LOAD DATA
PARAMETERS
CODE EN 1993-1-1:2005
NA 1
BEAM 3 ALL
DMAX 0.6 ALL
DMIN 0.3 ALL
UNL 0.3 ALL
SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
FINISH

STAAD Output File
**Application Examples**

**EX. British Design Examples**

11. 1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
12. 15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
13. 19 7 14 ; 20 2 7 ; 21 9 15
14. 22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
15. 26 17 19 ; 27 12 17 ; 28 5 12
16. MEMB PROP BRITISH
17. 1 TO 28 TABLE ST UB305X165X40
18. * MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
19. MEMB RELEASE
20. 1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
21. 4 9 13 14 15 18 27 25 19 21 TO 24 END MZ
22. UNIT MMS
23. DEFINE MATERIAL START
24. ISOTROPIC STEEL
25. E 210
26. POISSON 0.3
27. DENSITY 7.68191E-008
28. ALPHA 6E-006
29. DAMP 0.03
30. TYPE STEEL
31. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
32. END DEFINE MATERIAL
33. UNIT MMS KN
34. CONSTANS
35. MATERIAL STEEL ALL
36. UNIT METER KN
37. SUPPORT
38. 1 5 13 17 20 21 FIXED
A FLOOR FRAME DESIGN WITH AREA LOAD -- PAGE NO. 2
39. LOADING 1 14.5 KN/SQ.M. DL+LL
40. FLOOR LOAD
41. YRANGE -0.5 0.5 FLOAD -14.5
**NOTE** about Floor/OneWay Loads/Weights.
Please note that depending on the shape of the floor you may
have to break up the FLOOR/ONEWAY LOAD into multiple commands.
For details please refer to Technical Reference Manual
Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.
42. PERFORM ANALYSIS PRINT LOAD DATA

--- PROBLEM STATISTICS ---

<table>
<thead>
<tr>
<th>NUMBER OF JOINTS</th>
<th>20</th>
</tr>
</thead>
<tbody>
<tr>
<td>NUMBER OF MEMBERS</td>
<td>28</td>
</tr>
<tr>
<td>NUMBER OF PLATES</td>
<td>0</td>
</tr>
<tr>
<td>NUMBER OF SOLIDS</td>
<td>0</td>
</tr>
<tr>
<td>NUMBER OF SURFACES</td>
<td>0</td>
</tr>
<tr>
<td>NUMBER OF SUPPORTS</td>
<td>6</td>
</tr>
</tbody>
</table>

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 42
TOTAL LOAD COMBINATION CASES = 0 SO FAR.
A FLOOR FRAME DESIGN WITH AREA LOAD -- PAGE NO. 3
LOADING 1 14.5 KN/SQ.M. DL+LL

---

MEMBER LOAD - UNIT KN METE

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0637 GY</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.1912 GY</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.3186 GY</td>
<td>0.24</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.4460 GY</td>
<td>0.33</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.5735 GY</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.7009 GY</td>
<td>0.52</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Application Examples</td>
<td></td>
<td>STAAD.Pro 4651 User Manual</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---------------------</td>
<td>---</td>
<td>--------------------------</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.8284 GY 0.61</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.9558 GY 0.70</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.9558 GY 0.80</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.8284 GY 0.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.7009 GY 0.98</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.5735 GY 1.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.4460 GY 1.17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.3186 GY 1.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.1912 GY 1.35</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-0.0637 GY 1.44</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-10.8750 GY 0.75 3.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.0637 GY 0.06</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.1912 GY 0.15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.3186 GY 0.24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.4460 GY 0.33</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.5735 GY 0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.7009 GY 0.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.8284 GY 0.61</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.9558 GY 0.70</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.8284 GY 3.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.7009 GY 3.98</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.5735 GY 4.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.4460 GY 4.17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.3186 GY 4.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.1912 GY 4.35</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>-0.0637 GY 4.44</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.0637 GY 0.06</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1912 GY 0.15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.3186 GY 0.24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.4460 GY 0.33</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.5735 GY 0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.7009 GY 0.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.8284 GY 0.61</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.9558 GY 0.70</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.9558 GY 0.80</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.8284 GY 0.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.7009 GY 0.98</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.5735 GY 1.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-10.8750 GY 6.75 6.75</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.9558 GY 6.80</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.8284 GY 6.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.7009 GY 6.98</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.5735 GY 7.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Row</td>
<td>Data 1</td>
<td>Data 2</td>
<td>Data 3</td>
<td>Data 4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-----</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.4460 GY</td>
<td>7.17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.3186 GY</td>
<td>7.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.1912 GY</td>
<td>7.35</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.0637 GY</td>
<td>7.44</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.0637 GY</td>
<td>0.06</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.1912 GY</td>
<td>0.15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.3186 GY</td>
<td>0.24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.4460 GY</td>
<td>0.33</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.5735 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.7009 GY</td>
<td>0.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.8284 GY</td>
<td>0.61</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.9558 GY</td>
<td>0.70</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.9558 GY</td>
<td>0.80</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.8284 GY</td>
<td>0.89</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.7009 GY</td>
<td>0.98</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.5735 GY</td>
<td>1.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.4460 GY</td>
<td>1.17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.3186 GY</td>
<td>1.26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.1912 GY</td>
<td>1.35</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.0637 GY</td>
<td>1.44</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.0637 GY</td>
<td>0.06</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.1912 GY</td>
<td>0.15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.3186 GY</td>
<td>0.24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.4460 GY</td>
<td>0.33</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.5735 GY</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.7009 GY</td>
<td>0.52</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.8284 GY</td>
<td>0.61</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.9558 GY</td>
<td>0.70</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-10.8750 GY</td>
<td>0.75</td>
<td>2.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.9558 GY</td>
<td>2.30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.8284 GY</td>
<td>2.39</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.7009 GY</td>
<td>2.48</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.5735 GY</td>
<td>2.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.4460 GY</td>
<td>2.67</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.3186 GY</td>
<td>2.76</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.1912 GY</td>
<td>2.85</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.0637 GY</td>
<td>2.94</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-0.0637 GY</td>
<td>0.06</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-0.1912 GY</td>
<td>0.15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

A FLOOR FRAME DESIGN WITH AREA LOAD
<table>
<thead>
<tr>
<th>Layer</th>
<th>Load (kN)</th>
<th>Displacement (mm)</th>
<th>Rotation (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>-0.5735 GY</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.7009 GY</td>
<td>0.52</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.8284 GY</td>
<td>0.61</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.9558 GY</td>
<td>0.70</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-10.8750 GY</td>
<td>0.75  2.25</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.9558 GY</td>
<td>2.30</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.8284 GY</td>
<td>2.39</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.7009 GY</td>
<td>2.48</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.5735 GY</td>
<td>2.58</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.4460 GY</td>
<td>2.67</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.3186 GY</td>
<td>2.76</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.1912 GY</td>
<td>2.85</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>-0.0637 GY</td>
<td>2.94</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.0637 GY</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.1912 GY</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.3186 GY</td>
<td>0.24</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.4460 GY</td>
<td>0.33</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.5735 GY</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.7009 GY</td>
<td>0.52</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.8284 GY</td>
<td>0.61</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.9558 GY</td>
<td>0.70</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.9558 GY</td>
<td>0.80</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.8284 GY</td>
<td>0.89</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.7009 GY</td>
<td>0.98</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.5735 GY</td>
<td>1.08</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.4460 GY</td>
<td>1.17</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.3186 GY</td>
<td>1.26</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.1912 GY</td>
<td>1.35</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.0637 GY</td>
<td>1.44</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0637 GY</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1912 GY</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.3186 GY</td>
<td>0.24</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.4460 GY</td>
<td>0.33</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.5735 GY</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.7009 GY</td>
<td>0.52</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.8284 GY</td>
<td>0.61</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.9558 GY</td>
<td>0.70</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-10.8750 GY</td>
<td>0.75  2.25</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.9558 GY</td>
<td>2.30</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.8284 GY</td>
<td>2.39</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.7009 GY</td>
<td>2.48</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.5735 GY</td>
<td>2.58</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.4460 GY</td>
<td>2.67</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.3186 GY</td>
<td>2.76</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.1912 GY</td>
<td>2.85</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0637 GY</td>
<td>2.94</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.4248 GY</td>
<td>0.03</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.3682 GY</td>
<td>0.09</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.3115 GY</td>
<td>0.16</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.2549 GY</td>
<td>0.22</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.1982 GY</td>
<td>0.28</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.1416 GY</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.0850 GY</td>
<td>0.40</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-0.0283 GY</td>
<td>0.46</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-0.0637 GY</td>
<td>0.06</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-0.1912 GY</td>
<td>0.15</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-0.3186 GY</td>
<td>0.24</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. British Design Examples

<table>
<thead>
<tr>
<th>Floor</th>
<th>GY</th>
<th>Area Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>-0.4460</td>
<td>0.33</td>
</tr>
<tr>
<td>6</td>
<td>-0.5735</td>
<td>0.42</td>
</tr>
<tr>
<td>6</td>
<td>-0.7009</td>
<td>0.52</td>
</tr>
<tr>
<td>6</td>
<td>-0.8284</td>
<td>0.61</td>
</tr>
<tr>
<td>6</td>
<td>-0.9558</td>
<td>0.70</td>
</tr>
<tr>
<td>6</td>
<td>-0.3328</td>
<td>0.77</td>
</tr>
<tr>
<td>6</td>
<td>-0.3186</td>
<td>0.80</td>
</tr>
<tr>
<td>6</td>
<td>-0.3044</td>
<td>0.83</td>
</tr>
<tr>
<td>6</td>
<td>-0.2903</td>
<td>0.86</td>
</tr>
<tr>
<td>6</td>
<td>-0.2761</td>
<td>0.89</td>
</tr>
<tr>
<td>6</td>
<td>-0.2620</td>
<td>0.92</td>
</tr>
<tr>
<td>6</td>
<td>-0.2478</td>
<td>0.95</td>
</tr>
<tr>
<td>6</td>
<td>-0.2336</td>
<td>0.98</td>
</tr>
<tr>
<td>22</td>
<td>-0.0637</td>
<td>0.06</td>
</tr>
<tr>
<td>22</td>
<td>-0.1912</td>
<td>0.15</td>
</tr>
<tr>
<td>22</td>
<td>-0.3186</td>
<td>0.24</td>
</tr>
<tr>
<td>22</td>
<td>-0.4460</td>
<td>0.33</td>
</tr>
<tr>
<td>22</td>
<td>-0.5735</td>
<td>0.42</td>
</tr>
<tr>
<td>22</td>
<td>-0.7009</td>
<td>0.52</td>
</tr>
<tr>
<td>22</td>
<td>-0.8284</td>
<td>0.61</td>
</tr>
<tr>
<td>22</td>
<td>-0.9558</td>
<td>0.70</td>
</tr>
<tr>
<td>22</td>
<td>-10.8750</td>
<td>0.75</td>
</tr>
<tr>
<td>22</td>
<td>-0.9558</td>
<td>2.30</td>
</tr>
<tr>
<td>22</td>
<td>-0.8284</td>
<td>2.39</td>
</tr>
<tr>
<td>22</td>
<td>-0.7009</td>
<td>2.48</td>
</tr>
<tr>
<td>22</td>
<td>-0.5735</td>
<td>2.58</td>
</tr>
<tr>
<td>22</td>
<td>-0.4460</td>
<td>2.67</td>
</tr>
<tr>
<td>22</td>
<td>-0.3186</td>
<td>2.76</td>
</tr>
<tr>
<td>22</td>
<td>-0.1912</td>
<td>2.85</td>
</tr>
<tr>
<td>22</td>
<td>-0.0637</td>
<td>2.94</td>
</tr>
<tr>
<td>4</td>
<td>-0.0637</td>
<td>0.06</td>
</tr>
<tr>
<td>4</td>
<td>-0.1912</td>
<td>0.15</td>
</tr>
<tr>
<td>4</td>
<td>-0.3186</td>
<td>0.24</td>
</tr>
<tr>
<td>4</td>
<td>-0.4460</td>
<td>0.33</td>
</tr>
<tr>
<td>4</td>
<td>-0.5735</td>
<td>0.42</td>
</tr>
<tr>
<td>4</td>
<td>-0.7009</td>
<td>0.52</td>
</tr>
<tr>
<td>4</td>
<td>-0.8284</td>
<td>0.61</td>
</tr>
<tr>
<td>4</td>
<td>-0.9558</td>
<td>0.70</td>
</tr>
<tr>
<td>4</td>
<td>-0.9558</td>
<td>0.80</td>
</tr>
<tr>
<td>4</td>
<td>-0.8284</td>
<td>0.89</td>
</tr>
<tr>
<td>4</td>
<td>-0.7009</td>
<td>0.98</td>
</tr>
<tr>
<td>4</td>
<td>-0.5735</td>
<td>1.08</td>
</tr>
<tr>
<td>4</td>
<td>-0.4460</td>
<td>1.17</td>
</tr>
<tr>
<td>4</td>
<td>-0.3186</td>
<td>1.26</td>
</tr>
<tr>
<td>4</td>
<td>-0.1912</td>
<td>1.35</td>
</tr>
<tr>
<td>4</td>
<td>-0.0637</td>
<td>1.44</td>
</tr>
<tr>
<td>28</td>
<td>-0.0637</td>
<td>0.06</td>
</tr>
<tr>
<td>28</td>
<td>-0.1912</td>
<td>0.15</td>
</tr>
<tr>
<td>28</td>
<td>-0.3186</td>
<td>0.24</td>
</tr>
<tr>
<td>28</td>
<td>-0.4460</td>
<td>0.33</td>
</tr>
<tr>
<td>28</td>
<td>-0.5735</td>
<td>0.42</td>
</tr>
<tr>
<td>28</td>
<td>-0.7009</td>
<td>0.52</td>
</tr>
<tr>
<td>28</td>
<td>-0.8284</td>
<td>0.61</td>
</tr>
<tr>
<td>28</td>
<td>-0.9558</td>
<td>0.70</td>
</tr>
<tr>
<td>28</td>
<td>-10.8750</td>
<td>0.75</td>
</tr>
<tr>
<td>28</td>
<td>-0.9558</td>
<td>2.30</td>
</tr>
<tr>
<td>28</td>
<td>-0.8284</td>
<td>2.39</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>28</td>
<td>-0.7009 GY</td>
<td>2.48</td>
</tr>
<tr>
<td>28</td>
<td>-0.5735 GY</td>
<td>2.58</td>
</tr>
<tr>
<td>28</td>
<td>-0.4460 GY</td>
<td>2.67</td>
</tr>
<tr>
<td>28</td>
<td>-0.3186 GY</td>
<td>2.76</td>
</tr>
<tr>
<td>28</td>
<td>-0.1912 GY</td>
<td>2.85</td>
</tr>
<tr>
<td>28</td>
<td>-0.0637 GY</td>
<td>2.94</td>
</tr>
<tr>
<td>9</td>
<td>-0.2336 GY</td>
<td>0.02</td>
</tr>
<tr>
<td>9</td>
<td>-0.2478 GY</td>
<td>0.08</td>
</tr>
<tr>
<td>9</td>
<td>-0.2620 GY</td>
<td>0.08</td>
</tr>
<tr>
<td>9</td>
<td>-0.2761 GY</td>
<td>0.11</td>
</tr>
<tr>
<td>9</td>
<td>-0.2903 GY</td>
<td>0.14</td>
</tr>
<tr>
<td>9</td>
<td>-0.3044 GY</td>
<td>0.17</td>
</tr>
<tr>
<td>9</td>
<td>-0.3186 GY</td>
<td>0.20</td>
</tr>
<tr>
<td>9</td>
<td>-0.3328 GY</td>
<td>0.23</td>
</tr>
<tr>
<td>9</td>
<td>-0.3470 GY</td>
<td>0.26</td>
</tr>
<tr>
<td>9</td>
<td>-0.3612 GY</td>
<td>0.29</td>
</tr>
<tr>
<td>9</td>
<td>-0.3754 GY</td>
<td>0.32</td>
</tr>
<tr>
<td>9</td>
<td>-0.3896 GY</td>
<td>0.35</td>
</tr>
<tr>
<td>9</td>
<td>-0.4038 GY</td>
<td>0.38</td>
</tr>
<tr>
<td>9</td>
<td>-0.4180 GY</td>
<td>0.41</td>
</tr>
<tr>
<td>9</td>
<td>-0.4322 GY</td>
<td>0.44</td>
</tr>
<tr>
<td>9</td>
<td>-0.4464 GY</td>
<td>0.47</td>
</tr>
<tr>
<td>8</td>
<td>-0.0637 GY</td>
<td>0.06</td>
</tr>
<tr>
<td>8</td>
<td>-0.0850 GY</td>
<td>0.10</td>
</tr>
<tr>
<td>8</td>
<td>-0.1063 GY</td>
<td>0.14</td>
</tr>
<tr>
<td>8</td>
<td>-0.1276 GY</td>
<td>0.18</td>
</tr>
<tr>
<td>8</td>
<td>-0.1489 GY</td>
<td>0.22</td>
</tr>
<tr>
<td>8</td>
<td>-0.1702 GY</td>
<td>0.26</td>
</tr>
<tr>
<td>8</td>
<td>-0.1915 GY</td>
<td>0.30</td>
</tr>
<tr>
<td>8</td>
<td>-0.2128 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>8</td>
<td>-0.2341 GY</td>
<td>0.38</td>
</tr>
<tr>
<td>8</td>
<td>-0.2554 GY</td>
<td>0.42</td>
</tr>
<tr>
<td>8</td>
<td>-0.2767 GY</td>
<td>0.46</td>
</tr>
<tr>
<td>24</td>
<td>-0.8284 GY</td>
<td>0.61</td>
</tr>
<tr>
<td>24</td>
<td>-0.9558 GY</td>
<td>0.70</td>
</tr>
<tr>
<td>24</td>
<td>-10.8750 GY</td>
<td>0.75</td>
</tr>
<tr>
<td>24</td>
<td>-0.7009 GY</td>
<td>2.48</td>
</tr>
<tr>
<td>24</td>
<td>-0.5735 GY</td>
<td>2.58</td>
</tr>
<tr>
<td>24</td>
<td>-0.4460 GY</td>
<td>2.67</td>
</tr>
<tr>
<td>24</td>
<td>-0.3186 GY</td>
<td>2.76</td>
</tr>
<tr>
<td>24</td>
<td>-0.1912 GY</td>
<td>2.85</td>
</tr>
<tr>
<td>24</td>
<td>-0.0637 GY</td>
<td>2.94</td>
</tr>
<tr>
<td>21</td>
<td>-0.0283 GY</td>
<td>0.04</td>
</tr>
<tr>
<td>21</td>
<td>-0.0850 GY</td>
<td>0.10</td>
</tr>
<tr>
<td>21</td>
<td>-0.1416 GY</td>
<td>0.16</td>
</tr>
<tr>
<td>21</td>
<td>-0.2082 GY</td>
<td>0.22</td>
</tr>
<tr>
<td>21</td>
<td>-0.2649 GY</td>
<td>0.28</td>
</tr>
<tr>
<td>21</td>
<td>-0.3215 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>21</td>
<td>-0.3782 GY</td>
<td>0.40</td>
</tr>
<tr>
<td>21</td>
<td>-0.4348 GY</td>
<td>0.46</td>
</tr>
<tr>
<td>24</td>
<td>-0.9558 GY</td>
<td>2.30</td>
</tr>
<tr>
<td>24</td>
<td>-0.8284 GY</td>
<td>2.39</td>
</tr>
<tr>
<td>24</td>
<td>-0.7009 GY</td>
<td>2.48</td>
</tr>
<tr>
<td>24</td>
<td>-0.5735 GY</td>
<td>2.58</td>
</tr>
<tr>
<td>24</td>
<td>-0.4460 GY</td>
<td>2.67</td>
</tr>
<tr>
<td>24</td>
<td>-0.3186 GY</td>
<td>2.76</td>
</tr>
<tr>
<td>24</td>
<td>-0.1912 GY</td>
<td>2.85</td>
</tr>
<tr>
<td>24</td>
<td>-0.0637 GY</td>
<td>2.94</td>
</tr>
<tr>
<td>21</td>
<td>-0.2931 GY</td>
<td>0.19</td>
</tr>
<tr>
<td>21</td>
<td>-0.8794 GY</td>
<td>0.45</td>
</tr>
<tr>
<td>21</td>
<td>-1.4656 GY</td>
<td>0.73</td>
</tr>
<tr>
<td>21</td>
<td>-2.0519 GY</td>
<td>1.02</td>
</tr>
<tr>
<td>21</td>
<td>-2.6381 GY</td>
<td>1.30</td>
</tr>
<tr>
<td>21</td>
<td>-3.2244 GY</td>
<td>1.59</td>
</tr>
<tr>
<td>21</td>
<td>-3.8106 GY</td>
<td>1.88</td>
</tr>
<tr>
<td>21</td>
<td>-4.3969 GY</td>
<td>2.16</td>
</tr>
<tr>
<td>21</td>
<td>-4.2408 GY</td>
<td>2.44</td>
</tr>
<tr>
<td>21</td>
<td>-3.6754 GY</td>
<td>2.72</td>
</tr>
<tr>
<td>21</td>
<td>-3.1099 GY</td>
<td>3.00</td>
</tr>
<tr>
<td>21</td>
<td>-2.5445 GY</td>
<td>3.27</td>
</tr>
<tr>
<td>21</td>
<td>-1.9790 GY</td>
<td>3.55</td>
</tr>
<tr>
<td>21</td>
<td>-1.4136 GY</td>
<td>3.82</td>
</tr>
<tr>
<td>21</td>
<td>-0.8482 GY</td>
<td>4.10</td>
</tr>
<tr>
<td>21</td>
<td>-0.2827 GY</td>
<td>4.34</td>
</tr>
<tr>
<td>11</td>
<td>-0.2986 GY</td>
<td>0.09</td>
</tr>
<tr>
<td>11</td>
<td>-0.8958 GY</td>
<td>0.22</td>
</tr>
<tr>
<td>11</td>
<td>-1.4929 GY</td>
<td>0.36</td>
</tr>
<tr>
<td>11</td>
<td>-2.0901 GY</td>
<td>0.50</td>
</tr>
<tr>
<td>11</td>
<td>-2.6873 GY</td>
<td>0.64</td>
</tr>
<tr>
<td>11</td>
<td>-3.2845 GY</td>
<td>0.78</td>
</tr>
<tr>
<td>11</td>
<td>-3.8816 GY</td>
<td>0.92</td>
</tr>
<tr>
<td>11</td>
<td>-4.4788 GY</td>
<td>1.06</td>
</tr>
<tr>
<td>11</td>
<td>-3.4589 GY</td>
<td>1.18</td>
</tr>
<tr>
<td>11</td>
<td>-2.9908 GY</td>
<td>1.29</td>
</tr>
<tr>
<td>11</td>
<td>-2.5366 GY</td>
<td>1.40</td>
</tr>
<tr>
<td>11</td>
<td>-2.0785 GY</td>
<td>1.51</td>
</tr>
<tr>
<td>11</td>
<td>-1.6184 GY</td>
<td>1.62</td>
</tr>
<tr>
<td>11</td>
<td>-1.1583 GY</td>
<td>1.72</td>
</tr>
<tr>
<td>11</td>
<td>-0.6902 GY</td>
<td>1.83</td>
</tr>
<tr>
<td>11</td>
<td>-0.2301 GY</td>
<td>1.93</td>
</tr>
<tr>
<td>19</td>
<td>-0.2773 GY</td>
<td>0.18</td>
</tr>
<tr>
<td>19</td>
<td>-0.8318 GY</td>
<td>0.42</td>
</tr>
<tr>
<td>19</td>
<td>-1.3863 GY</td>
<td>0.69</td>
</tr>
<tr>
<td>19</td>
<td>-1.9408 GY</td>
<td>0.95</td>
</tr>
<tr>
<td>19</td>
<td>-2.4953 GY</td>
<td>1.22</td>
</tr>
<tr>
<td>19</td>
<td>-3.0499 GY</td>
<td>1.49</td>
</tr>
<tr>
<td>19</td>
<td>-3.6044 GY</td>
<td>1.76</td>
</tr>
<tr>
<td>19</td>
<td>-4.1589 GY</td>
<td>2.03</td>
</tr>
<tr>
<td>19</td>
<td>-4.7488 GY</td>
<td>2.31</td>
</tr>
<tr>
<td>19</td>
<td>-3.8816 GY</td>
<td>2.60</td>
</tr>
<tr>
<td>19</td>
<td>-3.2845 GY</td>
<td>2.89</td>
</tr>
<tr>
<td>19</td>
<td>-2.6873 GY</td>
<td>3.18</td>
</tr>
<tr>
<td>19</td>
<td>-2.0901 GY</td>
<td>3.47</td>
</tr>
<tr>
<td>19</td>
<td>-1.4929 GY</td>
<td>3.76</td>
</tr>
<tr>
<td>19</td>
<td>-0.8958 GY</td>
<td>4.05</td>
</tr>
<tr>
<td>19</td>
<td>-0.2986 GY</td>
<td>4.31</td>
</tr>
<tr>
<td>6</td>
<td>-2.6866 GY</td>
<td>0.06</td>
</tr>
<tr>
<td>6</td>
<td>-2.3284 GY</td>
<td>0.19</td>
</tr>
<tr>
<td>6</td>
<td>-1.9702 GY</td>
<td>0.31</td>
</tr>
<tr>
<td>6</td>
<td>-1.6128 GY</td>
<td>0.44</td>
</tr>
<tr>
<td>6</td>
<td>-1.2537 GY</td>
<td>0.56</td>
</tr>
<tr>
<td>6</td>
<td>-0.8955 GY</td>
<td>0.68</td>
</tr>
<tr>
<td>6</td>
<td>-0.5373 GY</td>
<td>0.81</td>
</tr>
<tr>
<td>6</td>
<td>-0.1791 GY</td>
<td>0.92</td>
</tr>
<tr>
<td>5</td>
<td>-0.2773 GY</td>
<td>0.09</td>
</tr>
<tr>
<td>5</td>
<td>-0.8318 GY</td>
<td>0.22</td>
</tr>
<tr>
<td>5</td>
<td>-1.3863 GY</td>
<td>0.36</td>
</tr>
<tr>
<td>5</td>
<td>-1.9408 GY</td>
<td>0.50</td>
</tr>
<tr>
<td>5</td>
<td>-2.4953 GY</td>
<td>0.64</td>
</tr>
<tr>
<td>5</td>
<td>-3.0499 GY</td>
<td>0.78</td>
</tr>
<tr>
<td>5</td>
<td>-3.6044 GY</td>
<td>0.92</td>
</tr>
<tr>
<td>5</td>
<td>-4.1589 GY</td>
<td>1.06</td>
</tr>
<tr>
<td>5</td>
<td>-1.4299 GY</td>
<td>1.15</td>
</tr>
<tr>
<td>5</td>
<td>-1.3808 GY</td>
<td>1.20</td>
</tr>
<tr>
<td>5</td>
<td>-1.3316 GY</td>
<td>1.25</td>
</tr>
<tr>
<td>5</td>
<td>-1.2825 GY</td>
<td>1.29</td>
</tr>
<tr>
<td>5</td>
<td>-1.2334 GY</td>
<td>1.34</td>
</tr>
<tr>
<td>5</td>
<td>-1.1842 GY</td>
<td>1.38</td>
</tr>
<tr>
<td>5</td>
<td>-1.1351 GY</td>
<td>1.43</td>
</tr>
<tr>
<td>5</td>
<td>-1.0859 GY</td>
<td>1.48</td>
</tr>
<tr>
<td>23</td>
<td>-0.1722 GY</td>
<td>0.20</td>
</tr>
<tr>
<td>23</td>
<td>-0.5166 GY</td>
<td>0.47</td>
</tr>
<tr>
<td>23</td>
<td>-0.8609 GY</td>
<td>0.76</td>
</tr>
<tr>
<td>23</td>
<td>-1.2053 GY</td>
<td>1.06</td>
</tr>
<tr>
<td>23</td>
<td>-1.5497 GY</td>
<td>1.36</td>
</tr>
<tr>
<td>23</td>
<td>-1.8941 GY</td>
<td>1.65</td>
</tr>
<tr>
<td>23</td>
<td>-2.2384 GY</td>
<td>1.95</td>
</tr>
<tr>
<td>23</td>
<td>-2.5828 GY</td>
<td>2.25</td>
</tr>
<tr>
<td>23</td>
<td>-2.2600 GY</td>
<td>2.53</td>
</tr>
<tr>
<td>23</td>
<td>-1.9586 GY</td>
<td>2.79</td>
</tr>
<tr>
<td>23</td>
<td>-1.6573 GY</td>
<td>3.05</td>
</tr>
<tr>
<td>23</td>
<td>-1.3560 GY</td>
<td>3.31</td>
</tr>
<tr>
<td>23</td>
<td>-1.0546 GY</td>
<td>3.58</td>
</tr>
<tr>
<td>23</td>
<td>-0.7533 GY</td>
<td>3.84</td>
</tr>
<tr>
<td>23</td>
<td>-0.4520 GY</td>
<td>4.09</td>
</tr>
<tr>
<td>23</td>
<td>-0.1507 GY</td>
<td>4.33</td>
</tr>
<tr>
<td>12</td>
<td>-0.2062 GY</td>
<td>0.87</td>
</tr>
<tr>
<td>12</td>
<td>-0.6185 GY</td>
<td>0.17</td>
</tr>
<tr>
<td>12</td>
<td>-1.0399 GY</td>
<td>0.27</td>
</tr>
<tr>
<td>12</td>
<td>-1.4432 GY</td>
<td>0.38</td>
</tr>
<tr>
<td>12</td>
<td>-1.8594 GY</td>
<td>0.49</td>
</tr>
<tr>
<td>12</td>
<td>-2.2679 GY</td>
<td>0.60</td>
</tr>
<tr>
<td>12</td>
<td>-2.6802 GY</td>
<td>0.71</td>
</tr>
<tr>
<td>12</td>
<td>-3.0926 GY</td>
<td>0.81</td>
</tr>
<tr>
<td>12</td>
<td>-2.2600 GY</td>
<td>0.91</td>
</tr>
<tr>
<td>12</td>
<td>-1.9586 GY</td>
<td>0.98</td>
</tr>
<tr>
<td>12</td>
<td>-1.6573 GY</td>
<td>1.06</td>
</tr>
<tr>
<td>12</td>
<td>-1.3560 GY</td>
<td>1.14</td>
</tr>
<tr>
<td>12</td>
<td>-1.0546 GY</td>
<td>1.22</td>
</tr>
<tr>
<td></td>
<td>-0.7533 GY</td>
<td>1.30</td>
</tr>
<tr>
<td></td>
<td>-0.4520 GY</td>
<td>1.38</td>
</tr>
<tr>
<td></td>
<td>-0.1507 GY</td>
<td>1.45</td>
</tr>
<tr>
<td></td>
<td>-0.1672 GY</td>
<td>0.28</td>
</tr>
<tr>
<td></td>
<td>-0.5016 GY</td>
<td>0.46</td>
</tr>
<tr>
<td></td>
<td>-0.8360 GY</td>
<td>0.74</td>
</tr>
<tr>
<td></td>
<td>-1.1704 GY</td>
<td>1.03</td>
</tr>
<tr>
<td></td>
<td>-1.5048 GY</td>
<td>1.32</td>
</tr>
<tr>
<td></td>
<td>-1.8392 GY</td>
<td>1.62</td>
</tr>
<tr>
<td></td>
<td>-2.1736 GY</td>
<td>1.91</td>
</tr>
<tr>
<td></td>
<td>-2.5080 GY</td>
<td>2.20</td>
</tr>
<tr>
<td></td>
<td>-2.3348 GY</td>
<td>2.48</td>
</tr>
<tr>
<td></td>
<td>-2.0235 GY</td>
<td>2.75</td>
</tr>
<tr>
<td></td>
<td>-1.7122 GY</td>
<td>3.02</td>
</tr>
<tr>
<td></td>
<td>-1.4089 GY</td>
<td>3.29</td>
</tr>
<tr>
<td></td>
<td>-1.0896 GY</td>
<td>3.57</td>
</tr>
<tr>
<td></td>
<td>-0.7783 GY</td>
<td>3.84</td>
</tr>
<tr>
<td></td>
<td>-0.4670 GY</td>
<td>4.10</td>
</tr>
<tr>
<td></td>
<td>-0.1557 GY</td>
<td>4.35</td>
</tr>
<tr>
<td></td>
<td>-1.6098 GY</td>
<td>0.83</td>
</tr>
<tr>
<td>8</td>
<td>-1.3951 GY</td>
<td>0.89</td>
</tr>
<tr>
<td>8</td>
<td>-1.1805 GY</td>
<td>0.16</td>
</tr>
<tr>
<td>8</td>
<td>-0.9659 GY</td>
<td>0.22</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>------------</td>
<td>------------</td>
</tr>
<tr>
<td>8</td>
<td>-0.7512 GY</td>
<td>0.28</td>
</tr>
<tr>
<td>8</td>
<td>-0.5366 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>8</td>
<td>-0.3220 GY</td>
<td>0.40</td>
</tr>
<tr>
<td>8</td>
<td>-0.1073 GY</td>
<td>0.46</td>
</tr>
<tr>
<td>7</td>
<td>-0.0997 GY</td>
<td>0.03</td>
</tr>
<tr>
<td>7</td>
<td>-0.2991 GY</td>
<td>0.07</td>
</tr>
<tr>
<td>7</td>
<td>-0.4984 GY</td>
<td>0.12</td>
</tr>
<tr>
<td>7</td>
<td>-0.6978 GY</td>
<td>0.16</td>
</tr>
<tr>
<td>7</td>
<td>-0.8972 GY</td>
<td>0.21</td>
</tr>
<tr>
<td>7</td>
<td>-1.0966 GY</td>
<td>0.25</td>
</tr>
<tr>
<td>7</td>
<td>-1.2959 GY</td>
<td>0.30</td>
</tr>
<tr>
<td>7</td>
<td>-1.4953 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>7</td>
<td>-0.5724 GY</td>
<td>0.37</td>
</tr>
<tr>
<td>7</td>
<td>-0.5571 GY</td>
<td>0.39</td>
</tr>
<tr>
<td>7</td>
<td>-0.5418 GY</td>
<td>0.41</td>
</tr>
<tr>
<td>7</td>
<td>-0.5266 GY</td>
<td>0.42</td>
</tr>
<tr>
<td>7</td>
<td>-0.5113 GY</td>
<td>0.44</td>
</tr>
<tr>
<td>7</td>
<td>-0.4961 GY</td>
<td>0.46</td>
</tr>
<tr>
<td>7</td>
<td>-0.4808 GY</td>
<td>0.47</td>
</tr>
<tr>
<td>7</td>
<td>-0.4655 GY</td>
<td>0.49</td>
</tr>
<tr>
<td>9</td>
<td>-0.0283 GY</td>
<td>0.04</td>
</tr>
<tr>
<td>9</td>
<td>-0.0850 GY</td>
<td>0.10</td>
</tr>
<tr>
<td>9</td>
<td>-0.1416 GY</td>
<td>0.16</td>
</tr>
<tr>
<td>9</td>
<td>-0.1982 GY</td>
<td>0.22</td>
</tr>
<tr>
<td>9</td>
<td>-0.2549 GY</td>
<td>0.28</td>
</tr>
<tr>
<td>9</td>
<td>-0.3115 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>9</td>
<td>-0.3682 GY</td>
<td>0.41</td>
</tr>
<tr>
<td>9</td>
<td>-0.4248 GY</td>
<td>0.47</td>
</tr>
<tr>
<td>9</td>
<td>-0.3682 GY</td>
<td>0.53</td>
</tr>
<tr>
<td>9</td>
<td>-0.3115 GY</td>
<td>0.59</td>
</tr>
<tr>
<td>9</td>
<td>-0.2549 GY</td>
<td>0.66</td>
</tr>
<tr>
<td>9</td>
<td>-0.1982 GY</td>
<td>0.72</td>
</tr>
<tr>
<td>9</td>
<td>-0.1982 GY</td>
<td>0.78</td>
</tr>
</tbody>
</table>

A FLOOR FRAME DESIGN WITH AREA LOAD
<table>
<thead>
<tr>
<th></th>
<th>-0.2549 GY</th>
<th>0.28</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>-0.3115 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>13</td>
<td>-0.3682 GY</td>
<td>0.41</td>
</tr>
<tr>
<td>13</td>
<td>-0.4248 GY</td>
<td>0.47</td>
</tr>
<tr>
<td>13</td>
<td>-0.4248 GY</td>
<td>0.53</td>
</tr>
<tr>
<td>13</td>
<td>-0.4248 GY</td>
<td>0.59</td>
</tr>
<tr>
<td>13</td>
<td>-0.3115 GY</td>
<td>0.66</td>
</tr>
<tr>
<td>13</td>
<td>-0.2549 GY</td>
<td>0.72</td>
</tr>
<tr>
<td>13</td>
<td>-0.1982 GY</td>
<td>0.78</td>
</tr>
<tr>
<td>13</td>
<td>-0.1416 GY</td>
<td>0.84</td>
</tr>
<tr>
<td>13</td>
<td>-0.0850 GY</td>
<td>0.90</td>
</tr>
<tr>
<td>13</td>
<td>-0.0283 GY</td>
<td>0.96</td>
</tr>
<tr>
<td>23</td>
<td>-0.0283 GY</td>
<td>0.98</td>
</tr>
<tr>
<td>23</td>
<td>-0.0850 GY</td>
<td>0.10</td>
</tr>
<tr>
<td>23</td>
<td>-0.1416 GY</td>
<td>0.16</td>
</tr>
<tr>
<td>23</td>
<td>-0.1982 GY</td>
<td>0.22</td>
</tr>
<tr>
<td>23</td>
<td>-0.2549 GY</td>
<td>0.28</td>
</tr>
<tr>
<td>23</td>
<td>-0.3115 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>23</td>
<td>-0.3682 GY</td>
<td>0.41</td>
</tr>
<tr>
<td>23</td>
<td>-0.4248 GY</td>
<td>0.47</td>
</tr>
<tr>
<td>23</td>
<td>-7.2500 GY</td>
<td>0.50</td>
</tr>
<tr>
<td></td>
<td>4.00</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>-0.4248 GY</td>
<td>4.03</td>
</tr>
<tr>
<td>23</td>
<td>-0.3682 GY</td>
<td>4.09</td>
</tr>
<tr>
<td>23</td>
<td>-0.3115 GY</td>
<td>4.16</td>
</tr>
<tr>
<td>23</td>
<td>-0.2549 GY</td>
<td>4.22</td>
</tr>
<tr>
<td>23</td>
<td>-0.1982 GY</td>
<td>4.28</td>
</tr>
<tr>
<td>23</td>
<td>-0.1416 GY</td>
<td>4.34</td>
</tr>
<tr>
<td>23</td>
<td>-0.0850 GY</td>
<td>4.40</td>
</tr>
<tr>
<td>23</td>
<td>-0.0283 GY</td>
<td>4.46</td>
</tr>
<tr>
<td>26</td>
<td>-0.0283 GY</td>
<td>0.84</td>
</tr>
<tr>
<td>26</td>
<td>-0.0850 GY</td>
<td>0.10</td>
</tr>
<tr>
<td>26</td>
<td>-0.1416 GY</td>
<td>0.16</td>
</tr>
<tr>
<td>26</td>
<td>-7.2500 GY</td>
<td>0.50</td>
</tr>
<tr>
<td>26</td>
<td>5.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.1982 GY</td>
<td>0.22</td>
</tr>
<tr>
<td>14</td>
<td>-0.2549 GY</td>
<td>0.28</td>
</tr>
<tr>
<td>14</td>
<td>-0.3115 GY</td>
<td>0.34</td>
</tr>
<tr>
<td>14</td>
<td>-0.3682 GY</td>
<td>0.41</td>
</tr>
<tr>
<td>14</td>
<td>-0.4248 GY</td>
<td>0.47</td>
</tr>
<tr>
<td>14</td>
<td>-0.4248 GY</td>
<td>5.53</td>
</tr>
<tr>
<td>14</td>
<td>-0.3115 GY</td>
<td>5.66</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. British Design Examples

<table>
<thead>
<tr>
<th>Layer</th>
<th>Load (kN/m)</th>
<th>Displacement (mm)</th>
<th>Strain (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>-0.2549 GY</td>
<td>5.72</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.1982 GY</td>
<td>5.78</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.1416 GY</td>
<td>5.90</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.0850 GY</td>
<td>5.96</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.0283 GY</td>
<td>0.04</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.0850 GY</td>
<td>0.10</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.1416 GY</td>
<td>0.16</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.1982 GY</td>
<td>0.22</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.2549 GY</td>
<td>0.28</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.3115 GY</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.3682 GY</td>
<td>0.41</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.4248 GY</td>
<td>0.47</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.4248 GY</td>
<td>0.53</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.3682 GY</td>
<td>0.59</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.3115 GY</td>
<td>0.66</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.2549 GY</td>
<td>0.72</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.1982 GY</td>
<td>0.78</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.1416 GY</td>
<td>0.84</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.0850 GY</td>
<td>0.90</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.0283 GY</td>
<td>0.96</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-7.2500 GY</td>
<td>0.00</td>
<td>0.50</td>
</tr>
<tr>
<td>13</td>
<td>-0.4248 GY</td>
<td>0.53</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.3682 GY</td>
<td>0.59</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.3115 GY</td>
<td>0.66</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.2549 GY</td>
<td>0.72</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.1982 GY</td>
<td>0.78</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.1416 GY</td>
<td>0.84</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0850 GY</td>
<td>0.90</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-0.0283 GY</td>
<td>0.96</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-7.2500 GY</td>
<td>0.50</td>
<td>1.50</td>
</tr>
<tr>
<td>10</td>
<td>-0.0283 GY</td>
<td>0.04</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.0850 GY</td>
<td>0.10</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1416 GY</td>
<td>0.16</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.1982 GY</td>
<td>0.22</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.2549 GY</td>
<td>0.28</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.3115 GY</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.3682 GY</td>
<td>0.41</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-0.4248 GY</td>
<td>0.47</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-7.2500 GY</td>
<td>0.00</td>
<td>2.00</td>
</tr>
<tr>
<td>11</td>
<td>-0.1133 GY</td>
<td>0.08</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-0.3398 GY</td>
<td>0.19</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-0.5664 GY</td>
<td>0.32</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-0.7930 GY</td>
<td>0.44</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-1.0195 GY</td>
<td>0.56</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-1.2461 GY</td>
<td>0.69</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-1.4727 GY</td>
<td>0.81</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-1.6992 GY</td>
<td>0.94</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-14.5000 GY</td>
<td>1.00</td>
<td>5.00</td>
</tr>
<tr>
<td>14</td>
<td>-1.6992 GY</td>
<td>5.06</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-1.4727 GY</td>
<td>5.19</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-1.2461 GY</td>
<td>5.31</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-1.0195 GY</td>
<td>5.44</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.7930 GY</td>
<td>5.56</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.5664 GY</td>
<td>5.68</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.3398 GY</td>
<td>5.81</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>-----</td>
<td>-----</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-0.1133 GY</td>
<td>5.92</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.1133 GY</td>
<td>0.88</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.3398 GY</td>
<td>0.19</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.5664 GY</td>
<td>0.32</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.7930 GY</td>
<td>0.44</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.0195 GY</td>
<td>0.56</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.2461 GY</td>
<td>0.69</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.4727 GY</td>
<td>0.81</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.6992 GY</td>
<td>0.94</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.6992 GY</td>
<td>1.06</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.4727 GY</td>
<td>1.19</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.2461 GY</td>
<td>1.31</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-1.0195 GY</td>
<td>1.44</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.7930 GY</td>
<td>1.56</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.5664 GY</td>
<td>1.68</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.3398 GY</td>
<td>1.81</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>-0.1133 GY</td>
<td>1.92</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-0.1133 GY</td>
<td>0.08</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-0.3398 GY</td>
<td>0.19</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-0.5664 GY</td>
<td>0.32</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-0.7930 GY</td>
<td>0.44</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.0195 GY</td>
<td>0.56</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.2461 GY</td>
<td>0.69</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.4727 GY</td>
<td>0.81</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.6992 GY</td>
<td>0.94</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>14.5000 GY</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.6992 GY</td>
<td>5.06</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.4727 GY</td>
<td>5.19</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.2461 GY</td>
<td>5.31</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-1.0195 GY</td>
<td>5.44</td>
<td></td>
</tr>
</tbody>
</table>

A FLOOR FRAME DESIGN WITH AREA LOAD

************ END OF DATA FROM INTERNAL STORAGE ************
### Application Examples

**EX. British Design Examples**

#### STEEL DESIGN

**STAAD.PRO MEMBER SELECTION - BS EN 1993-1-1:2005**

**NATIONAL ANNEX - NA to BS EN 1993-1-1:2005**

**PROGRAM CODE REVISION V1.13 BS_EC3_2005/1**

**A FLOOR FRAME DESIGN WITH AREA LOAD**

---

**ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>2 ST UB406X140X46 (BRITISH SECTIONS)</th>
<th>PASS EC-6.2.5</th>
<th>0.875</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6 ST UB406X140X39 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.782</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11 ST UB457X152X52 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.878</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>14 ST UB305X102X33 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.842</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>15 ST UB305X102X25 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.782</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>16 ST UB305X102X25 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.541</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>18 ST UB305X102X25 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.939</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19 ST UB457X152X60 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.956</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>21 ST UB305X102X25 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.540</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>23 ST UB305X102X25 (BRITISH SECTIONS)</td>
<td>PASS EC-6.2.5</td>
<td>0.418</td>
</tr>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**A FLOOR FRAME DESIGN WITH AREA LOAD**

---

**ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/</th>
<th>CRITICAL COND/</th>
<th>RATIO/</th>
<th>LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>MY</td>
<td>MZ</td>
<td>LOCATION</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>24 ST UB305X102X25 (BRITISH SECTIONS)</th>
<th>PASS EC-6.2.5</th>
<th>0.939</th>
</tr>
</thead>
</table>
**Warning** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
COMMANDS.

A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   17

*********** END OF THE STAAD.Pro RUN ***********
**** DATE= APR 14,2019   TIME= 22:53:33 ****
************************************************************
*   For technical assistance on STAAD.Pro, please visit    *
*   http://www.bentley.com/en/support/                     *
*   Details about additional assistance from               *
*   Bentley and Partners can be found at program menu      *
*   Help->Technical Support                                *
*   Copyright (c) 1997-2017 Bentley Systems, Inc.         *
*   http://www.bentley.com                                *
************************************************************

EX. UK-3 Soil Springs for Portal Frame

A portal frame type steel structure is sitting on concrete footings. The soil is to be considered as an elastic
foundation. The value of soil subgrade reaction is known from which spring constants are calculated by
multiplying the subgrade reaction by the tributary area of each modeled spring.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-3 Soil
Springs for Portal Frame.STD when you install the program.
Figure 485: Example Problem No. 3

Where:

\[ B = 6 \text{ m}, \quad H = 3 \text{ m}, \quad F_1 = 1.2 \text{ m}, \quad F_2 = 2.4 \text{ m} \]
\[ P = 20 \text{ KN}, \quad w = 4.5 \text{ KN/m} \]
Soil Subgrade Reaction = 41,666.67 KN/m³

Table 659: Spring constant calculation

<table>
<thead>
<tr>
<th>Springs of Joints</th>
<th>Spring Constant</th>
</tr>
</thead>
<tbody>
<tr>
<td>1, 5, 10, &amp; 14 (Edges)</td>
<td>2.4 x 0.3 x 41,666.67 = 30,000 KN/m</td>
</tr>
<tr>
<td>2, 3, 4, 11, 12, &amp; 13 (Interior)</td>
<td>2.4 x 0.6 x 41,666.67 = 60,000 KN/m</td>
</tr>
</tbody>
</table>

Actual input is shown in bold lettering followed by explanation.
Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

UNIT METER KNS

Defines the input units for the data that follows.

JOINT COORDINATES
1 0.0 0.0 0.0 5 2.4 0.0 0.0
6 1.2 3.0 0.0 ; 7 1.2 6.0 0.0
8 7.2 6.0 0.0 ; 9 7.2 3.0 0.0
10 6.0 0.0 0.0 14 8.4 0.0 0.0

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCES
1 1 2 4
5 3 6 ; 6 6 7
7 7 8 ; 8 6 9
9 8 9 ; 10 9 12
11 10 11 14

Defines the members by the joints to which they are connected.

MEMBER PROPERTIES BRITISH
1 4 11 14 PRIS YD 0.30 ZD 2.40
2 3 12 13 PRIS YD 0.60 ZD 2.40
5 6 9 10 TABLE ST J0254X203
7 8 TA ST UB305X165X40

The first two lines define member properties as prismatic (PRIS) followed by depth (YD) and width (ZD) values. The program will calculate the properties necessary to perform the analysis. See G.6.1 Prismatic Properties (on page 2323) for additional information. Member properties for the remaining members are chosen from the British steel table. The term ST stands for standard single section.

UNIT MMS
* E FOR STEEL IS 210 (KN/sq.mm.) AND FOR CONCRETE 21 (KN/sq.mm.)
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 76.977e-009
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 21
POISSON 0.17
DENSITY 23.534e-009
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
The CONSTANT command initiates input for material constants like modulus of elasticity, Density, and Poisson's ratio. The length unit is changed from METER to MM to facilitate the input in familiar units.

**Note:** Any line beginning with an asterisk (*) character is treated as a comment line.

The supports for the structure are specified above. The first set of joints are supports restrained in all directions except global moment-z (MZ). Also, a spring having a spring constant of 60 KN/mm is provided in the global Y direction at these nodes. The second set is similar to the former except for a different value of the spring constant.

Load case 1 is initiated followed by a title.

The selfweight of the structure is specified as acting in the global Y direction with a -1.0 factor.

Since global Y is vertically upwards, the -1.0 factor indicates that this load will act downwards.

Load 1 contains joint loads also. FX indicates that the load is a force in the global X direction.

Load 1 contains member loads also. GY indicates that the load acts in the global Y direction. The term UNI stands for uniformly distributed load, and is applied on members 7 and 8, acting downwards.

This command instructs the program to proceed with the analysis.

The above PRINT command instructs the program to print analysis results which include joint displacements, member forces and support reactions.

This command terminates the STAAD run.

**Input File**

STAAD PLANE PORTAL ON FOOTING FOUNDATION
UNIT METER KNS
JOINT COORDINATES
1 0.0 0.0 0.0 5 2.4 0.0 0.0
6 1.2 3.0 0.0 ; 7 1.2 6.0 0.0
8 7.2 6.0 0.0 ; 9 7.2 3.0 0.0
10 6.0 0.0 0.0 14 8.4 0.0 0.0
MEMBER INCIDENCES
MEMBER PROPERTIES BRITISH
1 4 11 14 PRIS YD 0.30 ZD 2.40
2 3 12 13 PRIS YD 0.60 ZD 2.40
5 6 9 10 TABLE ST J0254X203
7 8 TA ST UB305X165X40
UNIT MMS
* E FOR STEEL IS 210 (KN/sq.mm.) AND FOR CONCRETE 21 (KN/sq.mm.)
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 76.977e-009
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 21
POISSON 0.17
DENSITY 23.534e-009
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
MATERIAL STEEL MEMB 5 TO 10
SUPPORTS
2 TO 4 11 TO 13 FIXED BUT MZ KFY 60
1 5 10 14 FIXED BUT MZ KFY 30
UNIT METER
LOADING 1 DEAD AND WIND LOAD COMBINED
SELF Y -1.0
JOINT LOAD
6 7 FX 20.0
MEMBER LOAD
7 8 UNI GY -45.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH

STAAD Output File
1. STAAD PLANE PORTAL ON FOOTING FOUNDATION

INPUT FILE: UK-3 Soil Springs for Portal Frame.STD

2. UNIT METER KNS

3. JOINT COORDINATES

4. 1 0.0 0.0 0.0 5 2.4 0.0 0.0
5. 6 1.2 3.0 0.0 ; 7 1.2 6.0 0.0
6. 8 7.2 6.0 0.0 ; 9 7.2 3.0 0.0
7. 10 6.0 0.0 0.0 14 8.4 0.0 0.0

8. MEMBER INCIDENCES

9. 1 1 2 4
10. 5 3 6 ; 6 6 7
11. 7 7 8 ; 8 6 9
12. 9 8 9 ; 10 9 12
13. 11 10 11 14

14. MEMBER PROPERTIES BRITISH

15. 1 4 11 14 PRIS YD 0.30 ZD 2.40
16. 2 3 12 13 PRIS YD 0.60 ZD 2.40
17. 5 6 9 10 TABLE ST J0254X203
18. 7 8 TA ST UB305X165X40

19. UNIT MMS

20. * E FOR STEEL IS 210 (KN/SQ.MM.) AND FOR CONCRETE 21 (KN/SQ.MM.)

21. DEFINE MATERIAL START

22. ISOTROPIC STEEL

23. E 210
24. POISSON 0.3
25. DENSITY 76.977E-009
26. ALPHA 6E-006
27. DAMP 0.03

28. TYPE STEEL

29. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2

30. ISOTROPIC CONCRETE

31. E 21
32. POISSON 0.17
33. DENSITY 23.534E-009
34. ALPHA 5E-006
35. DAMP 0.05
36. G 9.25

37. TYPE CONCRETE

38. STRENGTH FCU 0.0275

39. PORTAL ON FOOTING FOUNDATION

-- PAGE NO. 2

40. END DEFINE MATERIAL

41. MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
42. MATERIAL STEEL MEMB 5 TO 10

43. SUPPORTS

44. 2 TO 4 11 TO 13 FIXED BUT MZ KFY 60
45. 1 5 10 14 FIXED BUT MZ KFY 30

46. UNIT METER

47. LOADING 1 DEAD AND WIND LOAD COMBINED

48. SELF Y -1.0
49. JOINT LOAD
50. 6 7 FX 20.0

51. MEMBER LOAD
52. 7 8 UNI GY -45.0
53. PERFORM ANALYSIS

PROBLEM STATISTICS

-----------------------------------
NUMBER OF JOINTS 14  NUMBER OF MEMBERS 14
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 10

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 32
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

54. PRINT ANALYSIS RESULTS

ANALYSIS RESULTS
PORTAL ON FOOTING FOUNDATION

JOINT DISPLACEMENT (CM  RADIANS)  STRUCTURE TYPE = PLANE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1075</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1232</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1367</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1466</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1531</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.5036</td>
<td>-0.1720</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0032</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>1.0714</td>
<td>-0.1898</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0045</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>1.0286</td>
<td>-0.2097</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0022</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0.5240</td>
<td>-0.1901</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0.0000</td>
<td>-0.0897</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0005</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1210</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0005</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1504</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0005</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1759</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0004</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>0.0000</td>
<td>-0.1969</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
</tbody>
</table>

PORTAL ON FOOTING FOUNDATION

SUPPORT REACTIONS -UNIT KNS  METE  STRUCTURE TYPE = PLANE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>73.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.09</td>
<td>82.00</td>
<td>0.00</td>
<td>0.00</td>
<td>16.30</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.00</td>
<td>87.96</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0.00</td>
<td>72.58</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>-40.09</td>
<td>90.24</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>16.30</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>0.00</td>
<td>105.56</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0.00</td>
<td>32.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00</td>
<td>45.92</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0.00</td>
<td>26.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>0.00</td>
<td>59.07</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

PORTAL ON FOOTING FOUNDATION

MEMBER END FORCES  STRUCTURE TYPE = PLANE

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>32.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>-22.08</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>16.30</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>96.83</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-16.30</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>4</td>
<td>-75.70</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>67.82</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>5</td>
<td>123.72</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>24.50</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>6</td>
<td>-35.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-24.50</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>7</td>
<td>45.92</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>8</td>
<td>162.09</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>24.82</td>
</tr>
</tbody>
</table>

All units are - UNIT KNS  METE (LOCAL)

STAAD.Pro 4669 User Manual
<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>-258.66</td>
<td>0.09</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-25.09</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>6</td>
<td>132.10</td>
<td>-56.74</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td>-129.67</td>
<td>56.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>96.64</td>
</tr>
<tr>
<td>8</td>
<td>-76.74</td>
<td>142.70</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-135.72</td>
</tr>
<tr>
<td>9</td>
<td>36.65</td>
<td>145.80</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-156.40</td>
</tr>
<tr>
<td>10</td>
<td>9</td>
<td>290.93</td>
<td>40.09</td>
<td>0.00</td>
<td>0.00</td>
<td>58.38</td>
</tr>
<tr>
<td>11</td>
<td>10</td>
<td>0.00</td>
<td>26.90</td>
<td>0.00</td>
<td>0.00</td>
<td>-13.09</td>
</tr>
<tr>
<td>12</td>
<td>11</td>
<td>0.00</td>
<td>89.32</td>
<td>0.00</td>
<td>0.00</td>
<td>-13.09</td>
</tr>
<tr>
<td>13</td>
<td>12</td>
<td>0.00</td>
<td>-134.13</td>
<td>0.00</td>
<td>0.00</td>
<td>-118.97</td>
</tr>
<tr>
<td>14</td>
<td>13</td>
<td>0.00</td>
<td>-48.90</td>
<td>0.00</td>
<td>0.00</td>
<td>-32.39</td>
</tr>
<tr>
<td>15</td>
<td>14</td>
<td>0.00</td>
<td>59.07</td>
<td>0.00</td>
<td>0.00</td>
<td>60.58</td>
</tr>
</tbody>
</table>

Related Links
- *M. To assign a spring support* (on page 814)
- *TR.27.1 Global Support Specification* (on page 2514)
- *Create Support dialog* (on page 2983)

**EX. UK-4 Inactive Members in a Braced Frame**

This example is a typical case of a load-dependent structure where the structural condition changes for different load cases. In this example, different bracing members are made inactive for different load cases. This is done to prevent these members from carrying any compressive forces.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-4 Inactive Members in a Braced Frame.STD when you install the program.
Where:

\[ L_1 = 4.5 \text{ m}, L_2 = 6.0 \text{ m} \]
\[ P_1 = 45 \text{ kN}, P_2 = 90 \text{ kN} \]

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE
* A PLANE FRAME STRUCTURE WITH TENSION BRACING

UNIT METER KNS
```

Defines the input units for the data that follows.

```
SET NL 3
```

This structure has to be analyzed for three primary load cases. Consequently, the modeling of our problem requires us to define three sets of data, with each set containing a load case and an associated analysis command. Also, the members which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have two commands, one called SET NL and the other called CHANGE. The SET NL command is used above to indicate the total number of
primary load cases that the file contains. The CHANGE command will come in later (after the PERFORM ANALYSIS command).

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0 0 0 3 12. 0. 0.</td>
</tr>
<tr>
<td>4 0 4.5 0 6 12. 4.5 0.</td>
</tr>
<tr>
<td>7 6. 9. 0. ; 8 12. 9. 0.</td>
</tr>
</tbody>
</table>

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7</td>
</tr>
<tr>
<td>8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6</td>
</tr>
<tr>
<td>13 6 7 ; 14 5 8</td>
</tr>
</tbody>
</table>

Defines the members by the joints to which they are connected.

<table>
<thead>
<tr>
<th>MEMBER TRUSS</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 TO 14</td>
</tr>
</tbody>
</table>

The preceding command defines that members 9 through 14 are of type truss. This means these members can only carry axial tension/compression and no moments.

<table>
<thead>
<tr>
<th>MEMBER PROP BRITISH</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 5 TABLE ST UB305X165X40</td>
</tr>
<tr>
<td>6 7 8 TA ST UB457X152X52</td>
</tr>
<tr>
<td>9 TO 14 TA LD UA150X150X10</td>
</tr>
</tbody>
</table>

Properties for all members are assigned from the British steel table. The word ST stands for standard single section. The word LD stands for long leg back-to-back double angle. Since the spacing between the two angles of the double angle is not provided, it is assumed to be 0.0.

<table>
<thead>
<tr>
<th>UNIT MMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td>E 210</td>
</tr>
<tr>
<td>POISSON 0.3</td>
</tr>
<tr>
<td>DENSITY 7.68191e-008</td>
</tr>
<tr>
<td>ALPHA 6e-006</td>
</tr>
<tr>
<td>DAMP 0.03</td>
</tr>
<tr>
<td>TYPE STEEL</td>
</tr>
<tr>
<td>STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2</td>
</tr>
<tr>
<td>END DEFINE MATERIAL</td>
</tr>
<tr>
<td>CONSTANTS</td>
</tr>
<tr>
<td>MATERIAL STEEL ALL</td>
</tr>
</tbody>
</table>

The DEFINE MATERIAL command is used to define a material. The CONSTANT command is used to assign this material all members. Length unit is changed from METER to MMS.

<table>
<thead>
<tr>
<th>SUPPORT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 2 3 PINNED</td>
</tr>
</tbody>
</table>

Pinned supports are specified at Joints 1, 2 and 3. The word PINNED signifies that no moments will be carried by these supports.

| INACTIVE MEMBERS 9 TO 14 |
The preceding command makes the listed members inactive. The stiffness contribution of these members will not be considered in the analysis till they are made active again.

```
UNIT METER
LOADING 1 DEAD AND LIVE LOAD
```

Load case 1 is initiated followed by a title. The length UNIT is changed from MMS to METER for input values which follow.

```
MEMBER LOAD
6 8 UNI GY -4.5
7 UNI GY -6.75
```

Load 1 contains member loads. GY indicates that the load acts in the global Y direction. The word UNI stands for uniformly distributed load. The load is applied on members 6, 7, and 8.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. It is worth noting that members 9 to 14 will not be used in this analysis since they were declared inactive earlier. In other words, for dead and live load, the bracing members are not used to carry any load.

```
CHANGES
```

The members inactivated earlier are restored using the CHANGE command.

```
INACTIVE MEMBERS 10 11 13
```

A new set of members are made inactive. The stiffness contribution from these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

```
LOADING 2 WIND FROM LEFT
```

Load case 2 is initiated followed by a title.

```
JOINT LOAD
4 FX 135 ; 7 FX 65
```

Load 2 contains joint loads. FX indicates that the load is a force in the global X direction. Nodes 4 and 7 are subjected to the loads.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. The analysis will be performed for load case 2 only.

```
CHANGE
```

The above CHANGE command is an instruction to re-activate all inactive members.

```
INACTIVE MEMBERS 9 12 14
```

Members 9, 12 and 14 are made inactive. The stiffness contribution of these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

```
LOADING 3 WIND FROM RIGHT
```

Load case 3 is initiated followed by a title.

```
JOINT LOAD
6 FX -135 ; 8 FX -65
```
Load 3 contains joint loads at nodes 6 and 8. FX indicates that the load is a force in the global X direction. The negative numbers (-135 and -65) indicate that the load is acting along the negative global X direction.

<table>
<thead>
<tr>
<th>LOAD COMBINATION 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0.75 2 0.75</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LOAD COMBINATION 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0.75 3 0.75</td>
</tr>
</tbody>
</table>

Load combination case 4 involves the algebraic summation of the results of load cases 1 and 2 after multiplying each by a factor of 0.75. For load combinations, the program simply gathers the results of the component primary cases, factors them appropriately, and combines them algebraically. Thus, an analysis in the real sense of the term (multiplying the inverted stiffness matrix by the load vector) is not carried out for load combination cases. Load combination case 5 combines the results of load cases 1 and 3.

**PERFORM ANALYSIS**

This command instructs the program to proceed with the analysis. Only primary load case 3 will be considered for this analysis. (As explained earlier, a combination case is not truly analyzed for, but handled using other means.)

**CHANGE**

The above **CHANGE** command will re-activate all inactive members.

**LOAD LIST ALL**

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the "active" load cases. The **LOAD LIST ALL** command enables all the load cases in the structure to be made active for further processing.

**PRINT MEMBER FORCES**

The preceding **PRINT** command is an instruction to produce a report, in the output file, of the member end forces.

**LOAD LIST 1 4 5**

A **LOAD LIST** command is a means of instructing the program to use only the listed load cases for further processing.

**PARAMETER**

<table>
<thead>
<tr>
<th>CODE</th>
<th>EN 1993-1-1:2005</th>
</tr>
</thead>
<tbody>
<tr>
<td>NA 1</td>
<td></td>
</tr>
<tr>
<td>BEAM</td>
<td>1 ALL</td>
</tr>
<tr>
<td>UNL</td>
<td>1.8 ALL</td>
</tr>
<tr>
<td>KY</td>
<td>0.5 ALL</td>
</tr>
</tbody>
</table>

The **PARAMETER** command is used to specify the steel design parameters (information on these parameters can be obtained from the manual where the implementation of the code is explained). The **BEAM** parameter is specified to perform design at every 1/12th point along the member length. **UNL** represents the unsupported length to be used for calculation of allowable bending stress. **KY 0.5 ALL** sets the effective length factor for column buckling about the local Y-axis to be 0.5 for ALL members.

**CHECK CODE ALL**

The above command instructs the program to perform a check to determine how you defined member sizes along with the latest analysis results meet the code requirements.

**FINISH**

This command terminates the STAAD run.
Input File

STAAD PLANE A PLANE FRAME STRUCTURE WITH
*TENSION BRACING
UNIT METER KNS
SET NL 3
JOINT COORDINATES
1 0 0 0 3 12. 0. 0.
4 0 4.5 0 6 12. 4.5 0.
7 6. 9. 0. ; 8 12. 9. 0.
MEMBER INCIDENCE
1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
8 7 8 ; 9 15 ; 10 2 4 ; 11 3 5 ; 12 2 6
13 6 7 ; 14 5 8
MEMBER TRUSS
9 TO 14
MEMBER PROP BRITISH
1 TO 5 TABLE ST UB305X165X40
6 7 8 TA ST UB457X152X52
9 TO 14 TA LD UA150X150X10
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORT
1 2 3 PINNED
INACTIVE MEMBERS 9 TO 14
UNIT METER
LOADING 1 DEAD AND LIVE LOAD
MEMBER LOAD
6 8 UNI GY -4.5
7 UNI GY -6.75
PERFORM ANALYSIS
CHANGES
INACTIVE MEMBERS 10 11 13
LOADING 2 WIND FROM LEFT
JOINT LOAD
4 FX 135 ; 7 FX 65
PERFORM ANALYSIS
CHANGE
INACTIVE MEMBERS 9 12 14
LOADING 3 WIND FROM RIGHT
JOINT LOAD
6 FX -135 ; 8 FX -65
LOAD COMBINATION 4
1 0.75 2 0.75
LOAD COMBINATION 5
1 0.75 3 0.75
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 4 5
PARAMETER
CODE EN 1993-1-1:2005
NA 1
BEAM 1 ALL
UNL 1.8 ALL
KY 0.5 ALL
CHECK CODE ALL
FINISH

STAAD Output File

***********
*           STAAD.Pro CONNECT Edition          *
*           Version  22.01.00.**                *
*           Proprietary Program of             *
*           Bentley Systems, Inc.              *
*           Date=    APR 14, 2019               *
*           Time=    22:54:35                   *
*                                               *
*  Licensed to: Bentley Systems Inc            *
*                                               *
                     ***************************************************************************
1. STAAD PLANE A PLANE FRAME STRUCTURE WITH
INPUT FILE: UK-4 Inactive Members in a Braced Frame.STD
2. * TENSION BRACING
3. UNIT METER KNS
4. SET NL 3
5. JOINT COORDINATES
6. 1 0 0 0 3 12. 0. 0.
7. 4 0 4.5 0 6 12. 4.5 0.
8. 7 6. 9. 0. ; 8 12. 9. 0.
9. MEMBER INCIDENCE
10. 1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
11. 8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
12. 13 6 7 ; 14 5 8
13. MEMBER TRUSS
14. 9 TO 14
15. MEMBER PROP BRITISH
16. 1 TO 5 TABLE ST UB305X165X40
17. 6 7 8 TA ST UB457X152X52
18. 9 TO 14 TA LD UA150X150X10
19. UNIT MMS
20. DEFINE MATERIAL START
21. ISOTROPIC STEEL
22. E 210
23. POISSON 0.3
24. DENSITY 7.68191E-008
25. ALPHA 6E-006
26. DAMP 0.03
27. TYPE STEEL
28. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
29. END DEFINE MATERIAL
30. CONSTANTS
31. MATERIAL STEEL ALL
32. SUPPORT
33. 1 2 3 PINNED
34. INACTIVE MEMBERS 9 TO 14
35. UNIT METER
36. LOADING 1 DEAD AND LIVE LOAD
37. MEMBER LOAD
38. 6 8 UNI GY -4.5
   A PLANE FRAME STRUCTURE WITH -- PAGE NO.  2
   * TENSION BRACING
39. 7 UNI GY -6.75
40. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          8  NUMBER OF MEMBERS      14
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

41. CHANGES
42. INACTIVE MEMBERS 10 11 13
43. LOADING 2 WIND FROM LEFT
44. JOINT LOAD
45. 4 FX 135 ; 7 FX 65
46. PERFORM ANALYSIS
47. CHANGE
48. INACTIVE MEMBERS 9 12 14
49. LOADING 3 WIND FROM RIGHT
50. JOINT LOAD
51. 6 FX -135 ; 8 FX -65
52. LOAD COMBINATION 4
53. 1  0.75 2  0.75
54. LOAD COMBINATION 5
55. 1  0.75 3  0.75
56. PERFORM ANALYSIS
57. CHANGE
58. LOAD LIST ALL
59. PRINT MEMBER FORCES

A PLANE FRAME STRUCTURE WITH -- PAGE NO.  3
   * TENSION BRACING
MEMBER END FORCES  STRUCTURE TYPE = PLANE
-----------------
ALL UNITS ARE -- KNS  METE     (LOCAL )

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>11.15</td>
<td>-0.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>-11.15</td>
<td>0.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-4.06</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>1</td>
<td>-1.07</td>
<td>0.78</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>1.07</td>
<td>-0.78</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>3.49</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>1</td>
<td>68.16</td>
<td>-0.68</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>-68.16</td>
<td>0.68</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-3.05</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>7.56</td>
<td>-0.09</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>-7.56</td>
<td>0.09</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.43</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>1</td>
<td>59.49</td>
<td>-1.18</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>-59.49</td>
<td>1.18</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-5.33</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. British Design Examples

| 2 | 1 | 2 | 51.94 | -0.07 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 1 | 2 | 46.95 | 0.54 | 0.00 | 0.00 | 0.00 | -0.00 |
|---|---|---|-------|-------|------|------|------|------|------|---|---|-------|-------|------|------|------|------|-----|
| 5 | -51.94 | 0.07 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 2.41 |
| 3 | 2 | 129.15 | -0.47 | 0.00 | 0.00 | 0.00 | 0.00 | -0.00 |
| 5 | -129.15 | 0.47 | 0.00 | 0.00 | 0.00 | 0.00 | -2.89 |
| 4 | 2 | 73.50 | 0.35 | 0.00 | 0.00 | 0.00 | 0.00 | -0.00 |
| 5 | -73.50 | -0.35 | 0.00 | 0.00 | 0.00 | 0.00 | 1.57 |
| 5 | 2 | 135.82 | -0.40 | 0.00 | 0.00 | 0.00 | 0.00 | -0.00 |
| 5 | -135.82 | 0.40 | 0.00 | 0.00 | 0.00 | 0.00 | -1.81 |
| 3 | 1 | 13.68 | -2.95 | 0.00 | 0.00 | 0.00 | 0.00 | -5.29 |
| 7 | -13.68 | 2.95 | 0.00 | 0.00 | 0.00 | 0.00 | -8.01 |
| 2 | 6 | 47.69 | 1.28 | 0.00 | 0.00 | 0.00 | 0.00 | -2.92 |
| 8 | -47.69 | -1.28 | 0.00 | 0.00 | 0.00 | 0.00 | -2.86 |
| 3 | 6 | -1.29 | -1.13 | 0.00 | 0.00 | 0.00 | 0.00 | -2.15 |
| 8 | 1.29 | 1.13 | 0.00 | 0.00 | 0.00 | 0.00 | -2.92 |
| 4 | 6 | 45.76 | 3.18 | 0.00 | 0.00 | 0.00 | 0.00 | -6.94 |
| 8 | -45.76 | -3.18 | 0.00 | 0.00 | 0.00 | 0.00 | -7.37 |
| 5 | 6 | 9.02 | 1.37 | 0.00 | 0.00 | 0.00 | 0.00 | 3.18 |
| 8 | -9.02 | -1.37 | 0.00 | 0.00 | 0.00 | 0.00 | 2.99 |
| 6 | 1 | 0.90 | 11.15 | 0.00 | 0.00 | 0.00 | 0.00 | 4.06 |
| 5 | -0.90 | 15.85 | 0.00 | 0.00 | 0.00 | 0.00 | -18.14 |
| 2 | 4 | 134.22 | -1.07 | 0.00 | 0.00 | 0.00 | 0.00 | -3.49 |
| 5 | -134.22 | 1.07 | 0.00 | 0.00 | 0.00 | 0.00 | -2.96 |
| 3 | 4 | 89.65 | 1.43 | 0.00 | 0.00 | 0.00 | 0.00 | 3.05 |
| 5 | -89.65 | -1.43 | 0.00 | 0.00 | 0.00 | 0.00 | 5.55 |
| 4 | 4 | 101.34 | 7.56 | 0.00 | 0.00 | 0.00 | 0.00 | 0.43 |
| 5 | -101.34 | 12.69 | 0.00 | 0.00 | 0.00 | 0.00 | -15.82 |
| 5 | 4 | 67.91 | 9.44 | 0.00 | 0.00 | 0.00 | 0.00 | 5.33 |
| 5 | -67.91 | 10.81 | 0.00 | 0.00 | 0.00 | 0.00 | -9.44 |
| 7 | 1 | -1.98 | 22.41 | 0.00 | 0.00 | 0.00 | 0.00 | 23.75 |
| 6 | 1.98 | 18.09 | 0.00 | 0.00 | 0.00 | 0.00 | -10.77 |

A PLANE FRAME STRUCTURE WITH * TENSION BRACING

MEMBER END FORCES STRUCTURE TYPE = PLANE

---

**Application Examples**

**EX. British Design Examples**

---

**STAAD.Pro 4678 User Manual**
### Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>5</td>
<td>72.35</td>
<td>-1.12</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2.94</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>196.97</td>
<td>1.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2.71</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>-196.97</td>
<td>-1.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>4.81</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>146.24</td>
<td>17.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>19.84</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-146.24</td>
<td>12.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-4.47</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>5</td>
<td>146.24</td>
<td>17.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>19.84</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-146.24</td>
<td>12.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-4.47</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>52.78</td>
<td>15.97</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>15.61</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>-52.78</td>
<td>14.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-10.92</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>3</td>
<td>136.62</td>
<td>1.29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>4.82</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>-136.62</td>
<td>-1.29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2.92</td>
<td></td>
</tr>
</tbody>
</table>

---

**A PLANE FRAME STRUCTURE WITH TENSION BRACING**

**MEMBER END FORCES STRUCTURE TYPE = PLANE**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
<td>-156.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>156.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-117.24</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>117.24</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>-91.79</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>91.79</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>3</td>
<td>-131.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>3</td>
<td>131.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>2</td>
<td>-83.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>2</td>
<td>83.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

---

**ALL UNITS ARE KNS METE (LOCAL)**

---

*STAAD.Pro 4679 User Manual*
Application Examples
EX. British Design Examples

3 2 0.00 0.00 0.00 0.00 0.00 0.00
6 0.00 0.00 0.00 0.00 0.00 0.00
4 2 -68.84 0.00 0.00 0.00 0.00 0.00
6 68.84 0.00 0.00 0.00 0.00 0.00
5 2 0.00 0.00 0.00 0.00 0.00 0.00
6 0.00 0.00 0.00 0.00 0.00 0.00
13 1 6 0.00 0.00 0.00 0.00 0.00 0.00
7 0.00 0.00 0.00 0.00 0.00 0.00
2 6 0.00 0.00 0.00 0.00 0.00 0.00
7 0.00 0.00 0.00 0.00 0.00 0.00

A PLANE FRAME STRUCTURE WITH * TENSION BRACING -- PAGE NO. 6
MEMBER END FORCES STRUCTURE TYPE = PLANE
---------------------------------------------------------
ALL UNITS ARE -- KNS METE (LOCAL )

MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
3 6 -76.79 0.00 0.00 0.00 0.00 0.00
7 76.79 0.00 0.00 0.00 0.00 0.00
4 6 0.00 0.00 0.00 0.00 0.00 0.00
7 0.00 0.00 0.00 0.00 0.00 0.00
5 6 -57.59 0.00 0.00 0.00 0.00 0.00
7 57.59 0.00 0.00 0.00 0.00 0.00
14 1 5 0.00 0.00 0.00 0.00 0.00 0.00
8 0.00 0.00 0.00 0.00 0.00 0.00
2 5 -77.83 0.00 0.00 0.00 0.00 0.00
8 77.83 0.00 0.00 0.00 0.00 0.00
3 5 0.00 0.00 0.00 0.00 0.00 0.00
8 0.00 0.00 0.00 0.00 0.00 0.00
4 5 -58.38 0.00 0.00 0.00 0.00 0.00
8 58.38 0.00 0.00 0.00 0.00 0.00
5 5 0.00 0.00 0.00 0.00 0.00 0.00
8 0.00 0.00 0.00 0.00 0.00 0.00

A PLANE FRAME STRUCTURE WITH * TENSION BRACING -- PAGE NO. 7
MEMBER END FORCES STRUCTURE TYPE = PLANE
---------------------------------------------------------
ALL UNITS ARE -- KNS METE (LOCAL )

MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
3 6 -76.79 0.00 0.00 0.00 0.00 0.00
7 76.79 0.00 0.00 0.00 0.00 0.00
4 6 0.00 0.00 0.00 0.00 0.00 0.00
7 0.00 0.00 0.00 0.00 0.00 0.00
5 6 -57.59 0.00 0.00 0.00 0.00 0.00
7 57.59 0.00 0.00 0.00 0.00 0.00
14 1 5 0.00 0.00 0.00 0.00 0.00 0.00
8 0.00 0.00 0.00 0.00 0.00 0.00
2 5 -77.83 0.00 0.00 0.00 0.00 0.00
8 77.83 0.00 0.00 0.00 0.00 0.00
3 5 0.00 0.00 0.00 0.00 0.00 0.00
8 0.00 0.00 0.00 0.00 0.00 0.00
4 5 -58.38 0.00 0.00 0.00 0.00 0.00
8 58.38 0.00 0.00 0.00 0.00 0.00
5 5 0.00 0.00 0.00 0.00 0.00 0.00
8 0.00 0.00 0.00 0.00 0.00 0.00

********************************************
NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
PROGRAM CODE REVISION V1.13 BS_EC3_2005/1
STEAD.PRO CODE CHECKING - BS EN 1993-1-1:2005
******************************************************

** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
DESIGN MIGHT NOT PROVIDE THE WORST CASE

1ST UB305X165X40 (BRITISH SECTIONS)
PASS EC-6.3.3-662 0.097
**Application Examples**  
*EX. British Design Examples*

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>59.49</th>
<th>-5.33</th>
<th>4.50</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2 ST UB305X165X40 (BRITISH SECTIONS) PASS</td>
<td>EC-6.3.3-662</td>
<td>0.149</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>135.82</th>
<th>-1.81</th>
<th>4.50</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3 ST UB305X165X40 (BRITISH SECTIONS) PASS</td>
<td>EC-6.3.3-662</td>
<td>0.112</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>45.79</th>
<th>-9.62</th>
<th>4.50</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4 ST UB305X165X40 (BRITISH SECTIONS) PASS</td>
<td>EC-6.3.3-662</td>
<td>0.129</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>101.47</th>
<th>3.98</th>
<th>4.50</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 5</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5 ST UB305X165X40 (BRITISH SECTIONS) PASS</td>
<td>EC-6.3.3-662</td>
<td>0.096</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>45.76</th>
<th>7.37</th>
<th>4.50</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 6</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6 ST UB457X152X52 (BRITISH SECTIONS) PASS</td>
<td>EC-6.3.3-662</td>
<td>0.176</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>101.34</th>
<th>-15.82</th>
<th>6.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 7</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7 ST UB457X152X52 (BRITISH SECTIONS) PASS</td>
<td>EC-6.3.3-662</td>
<td>0.241</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>146.24</th>
<th>19.84</th>
<th>0.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8 ST UB457X152X52 (BRITISH SECTIONS) PASS</td>
<td>EC-6.3.3-662</td>
<td>0.096</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>C</th>
<th>0.00</th>
<th>50.12</th>
<th>9.62</th>
<th>0.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>*** WARNING: BEAM PARAM CHANGED FROM DEFAULT OF 3 DESIGN MIGHT NOT PROVIDE THE WORST CASE UTILIZATION FOR MEMBER 9</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9 LD UA150X150X10 (BRITISH SECTIONS) PASS</td>
<td>EC-6.2.3 (T)</td>
<td>0.085</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

* TENSION BRACING

A PLANE FRAME STRUCTURE WITH **PAGE NO. 8**
117.24 T  0.00  0.00  0.00
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
DESIGN MIGHT NOT PROVIDE THE WORST CASE
UTILIZATION FOR MEMBER  10
   10 LD UA150X150X10 (BRITISH SECTIONS)
   PASS EC-6.2.3 (T)  0.061
5

83.41 T  0.00  0.00  0.00
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
DESIGN MIGHT NOT PROVIDE THE WORST CASE
UTILIZATION FOR MEMBER  11
   11 LD UA150X150X10 (BRITISH SECTIONS)
   PASS EC-6.2.3 (T)  0.074
5

102.46 T  0.00  0.00  0.00
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
DESIGN MIGHT NOT PROVIDE THE WORST CASE
UTILIZATION FOR MEMBER  12
   12 LD UA150X150X10 (BRITISH SECTIONS)
   PASS EC-6.2.3 (T)  0.050
4

A PLANE FRAME STRUCTURE WITH -- PAGE NO.  9
* TENSION BRACING

68.84 T  0.00  0.00  0.00
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
DESIGN MIGHT NOT PROVIDE THE WORST CASE
UTILIZATION FOR MEMBER  13
   13 LD UA150X150X10 (BRITISH SECTIONS)
   PASS EC-6.2.3 (T)  0.042
5

57.59 T  0.00  0.00  0.00
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
DESIGN MIGHT NOT PROVIDE THE WORST CASE
UTILIZATION FOR MEMBER  14
   14 LD UA150X150X10 (BRITISH SECTIONS)
   PASS EC-6.2.3 (T)  0.042
4

58.38 T  0.00  0.00  0.00
*************** END OF TABULATED RESULT OF DESIGN ***************

68. FINISH
*************** END OF THE STAAD.Pro RUN ***************
**** DATE= APR 14,2019   TIME= 22:54:36 ****
A PLANE FRAME STRUCTURE WITH -- PAGE NO. 10
* TENSION BRACING

Related Links
- Member Specification dialog (on page 2962)
EX. UK-5 Support Settlement on a Portal Frame

This example demonstrates the application of support displacement load (also known as a sinking support) on a space frame structure.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-5 Support Settlement on a Portal Frame.STD when you install the program.

![Diagram of portal frame structure](image)

*Figure 487: Example Problem No. 5*

Where:

L1 = 3.0m  
L2 = 6.0m

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNITS METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 3.0 0.0
3 6.0 3.0 0.0 ; 4 6.0 0.0 0.0
5 6.0 3.0 6.0 ; 6 6.0 0.0 6.0
```

Joint number followed by X, Y and Z coordinates are provided above.
**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 2 3</td>
</tr>
<tr>
<td>4 5 6</td>
</tr>
</tbody>
</table>

Defines the members by the joints to which they are connected.

<table>
<thead>
<tr>
<th>UNIT MMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEMB PROP</td>
</tr>
<tr>
<td>1 TO 5 PRIS AX 6450 IZ 1.249E+08 IY 1.249E+08 IX 4.162E+06</td>
</tr>
</tbody>
</table>

Member properties have been defined above using the PRISMATIC attribute. Values of AX (area), IZ (moment of inertia about major axis), IY (moment of inertia about minor axis) and IX (torsional constant) are provided in MMS unit.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210.
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

Material are defined using the DEFINE MATERIAL command and then assigned using the CONSTANT command.

<table>
<thead>
<tr>
<th>SUPPORT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 4 6 FIXED</td>
</tr>
</tbody>
</table>

Joints 1, 4 and 6 are fixed supports.

<table>
<thead>
<tr>
<th>LOAD</th>
<th>SUPPORT DISPLACEMENT LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 FY -15</td>
<td></td>
</tr>
</tbody>
</table>

Load 1 is a support displacement load which is also commonly known as a sinking support. FY signifies that the support settlement is in the global Y direction and the value of this settlement is 15mm downward.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
PRINT ANALYSIS RESULTS
```

The above PRINT command instructs the program to print joint displacements, support reactions and member forces.

```
FINISH
```

This command terminates the STAAD run.

**Input File**

```
STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
UNITS METER KNS
```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 3.0 0.0
3 6.0 3.0 0.0 ; 4 6.0 0.0 0.0
5 6.0 3.0 6.0 ; 6 6.0 0.0 6.0

MEMBER INCIDENCE
1 1 2 3
4 3 5 ; 5 5 6

UNIT MMS
MEMB PROP
1 TO 5 PRIS AX 6450 IZ 1.249E+08 IY 1.249E+08 IX 4.162E+06

DEFINE MATERIAL START
ISOTROPIC STEEL
E 210.
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL

CONSTANTS
MATERIAL STEEL ALL

SUPPORT
1 4 6 FIXED
LOADING 1 SINKING SUPPORT
SUPPORT DISPLACEMENT LOAD
4 FY -15.

PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH
13. DEFINE MATERIAL START
14. ISOTROPIC STEEL
16. POISSON 0.3
17. DENSITY 7.68191E-008
18. ALPHA 6E-006
19. DAMP 0.03
20. TYPE STEEL
21. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
22. END DEFINE MATERIAL
23. CONSTANTS
24. MATERIAL STEEL ALL
25. SUPPORT
26. 1 4 6 FIXED
27. LOADING 1 SINKING SUPPORT
28. SUPPORT DISPLACEMENT LOAD
29. 4 FY -15.
30. PERFORM ANALYSIS

TEST FOR SUPPORT DISPLACEMENT

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

ANALYSIS RESULTS
TEST FOR SUPPORT DISPLACEMENT

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE
------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
2    1     0.2737   -0.0012   -0.0323   -0.0002    0.0006   -0.0019
3    1     0.2735   -1.4975   -0.2735   -0.0018    0.0000   -0.0018
4    1     0.0000   -1.5000    0.0000    0.0000    0.0000    0.0000
5    1     0.0323   -0.0012   -0.2737   -0.0019   -0.0006   -0.0002
6    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000

SUPPORT REACTIONS -UNIT KNS  MMS STRUCTURE TYPE = SPACE
-----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z   MOM-X   MOM-Y   MOM Z
1    1      0.46      5.63     -0.46     67.13  15492.89
4    1     -11.26     -0.43      0.43   15509.16   0.00  15509.16
6    1     -0.90      5.63     -0.46   15492.89   67.13   2780.50

MEMBER END FORCES STRUCTURE TYPE = SPACE
----------------------------------------
MEMBER  LOAD  JT   AXIAL   SHEAR-Y   SHEAR-Z   TORSION   MOM-Y   MOM-Z
1    1     5.63    -0.46     0.90    -67.13  -2780.50  15492.89
2    1     5.63    -0.46     0.90    67.13   94.34  -16878.26
3    1     5.63    -0.46     0.90   94.34  -16878.26  -5305.18  16904.29
3    1    -11.26     -0.43      0.43   -94.34 -16809.94  -16809.94
4    11.26     0.43    -0.43     0.00  15509.16  15509.16

STAAD.Pro 4686 User Manual
EX. UK-6 Prestress and Poststress Loading

This is an example of prestress loading in a plane frame structure.

It covers two situations:

1. From the member on which it is applied, the prestressing effect is transmitted to the rest of the structure through the connecting members (known in the program as PRESTRESS load).

2. The prestressing effect is experienced by the member(s) alone and not transmitted to the rest of the structure (known in the program as POSTSTRESS load).

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-6 Prestress and Poststress Loading.STD when you install the program.
Figure 488: Example Problem No. 6

Where:

- $L_1 = 4.5$ m
- $L_2 = 6$ m
- $L_3 = 12$ m

Actual input is shown in bold lettering followed by explanation.

**STAAD PLANE FRAME WITH PRESTRESSING LOAD**

Every input has to start with the term **STAAD**. The term **PLANE** signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

**UNIT METER KNS**

Defines the input units for the data that follows.

**JOINT COORD**

```
1 0. 0. ; 2 12. 0. ; 3 0. 6. ; 4 12. 6.
5 0. 10.5 ; 6 12. 10.5 ; 7 0. 15. ; 8 12. 15.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

**Note**: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

**MEMBER INCIDENCE**

```
1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
```

---

*Application Examples*

**EX. British Design Examples**

---

*STAAD.Pro 4688 User Manual*
Defines the members by the joints to which they are connected.

| SUPPORT | 1 2 FIXED |

The supports at joints 1 and 2 are defined to be fixed supports.

| MEMB PROP | 1 TO 9 PRI AX 0.2044 IZ 8.631E-03 |

Member properties are provided using the PRI (prismatic) attribute. Values of area (AX) and moment of inertia about the major axis (IZ) are provided.

| UNIT MMS |
| DEFINE MATERIAL START |
| ISOTROPIC CONCRETE |
| E 21.0 |
| POISSON 0.17 |
| DENSITY 2.36158e-008 |
| ALPHA 5e-006 |
| DAMP 0.05 |
| G 9.25 |
| TYPE CONCRETE |
| STRENGTH FCU 0.0275 |
| END DEFINE MATERIAL |
| CONSTANTS |
| MATERIAL CONCRETE ALL |

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. Length unit is changed from METER to MMS to facilitate the input.

| LOADING 1 PRESTRESSING LOAD |
| MEMBER PRESTRESS |
| 7 8 FORCE 1350. ES 75. EM -300. EE 75. |

Load case 1 is initiated followed by a title. Load 1 contains PRESTRESS load. Members 7 and 8 have a cable force of 1350 KNs. The location of the cable at the start (ES) and end (EE) is 75 MMs above the centre of gravity while at the middle (EM) it is 300 MMs below the c.g. The assumptions and facts associated with this type of loading are explained in section 1 of the Technical Reference Manual.

| LOADING 2 POSTSTRESSING LOAD |
| MEMBER POSTSTRESS |
| 7 8 FORCE 1350. ES 75. EM -300. EE 75. |

Load case 2 is initiated followed by a title. Load 2 is a POSTSTRESS load. Members 7 and 8 have a cable force of 1350 KNs. The location of the cable is the same as in load case 1. For a difference between PRESTRESS loading and POSTSTRESS loading, as well as additional information about both types of loads, please refer to section 1 of the Technical Reference Manual.

| PERFORM ANALYSIS |

This command instructs the program to perform the analysis.

| UNIT METER |
| PRINT ANALYSIS RESULTS |

The preceding command is an instruction to write joint displacements, support reactions and member forces in the output file. The preceding line causes the results to be written in the length unit of meters.

| FINISH |

This command terminates the STAAD run.
Input File

STAAD PLANE FRAME WITH PRESTRESSING LOAD
UNIT METER KNS
JOINT COORD
1 0. 0. ; 2 12. 0. ; 3 0. 6. ; 4 12. 6.
5 0. 10.5 ; 6 12. 10.5 ; 7 0. 15. ; 8 12. 15.
MEMBER INCIDENCE
1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
SUPPORT
1 2 FIXED
MEMB PROP
1 TO 9 PRI AX 0.2044 IZ 8.631E-03
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
LOADING 1 PRESTRESSING LOAD
MEMBER PRESTRESS
7 8 FORCE 1350. ES 75. EM -300. EE 75.
LOADING 2 POSTSTRESSING LOAD
MEMBER POSTSTRESS
7 8 FORCE 1350. ES 75. EM -300. EE 75.
PERFORM ANALYSIS
UNIT METER
PRINT ANALYSIS RESULTS
FINISH

STAAD Output File

**************************************************************************
*                    STAAD.Pro CONNECT Edition                      *
*                  Version 22.01.00.**                             *
*          Proprietary Program of                                *
*     Bentley Systems, Inc.                                     *
*            Date=    APR 14, 2019                              *
*            Time=    22:54:46                                  *
*                                                      *
* Licensed to: Bentley Systems Inc                          *
**************************************************************************

1. STAAD PLANE FRAME WITH PRESTRESSING LOAD
INPUT FILE: UK-6 Prestress and Poststress Loading.STD
2. UNIT METER KNS
3. JOINT COORD
4. MEMBER INCIDENCE
5. 1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6. 6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
7. SUPPORT
8. 1 2 FIXED
9. MEMB PROP
10. 1 TO 9 PRI AX 0.2044 IZ 8.631E-03
11. UNIT MMS
12. DEFINE MATERIAL START
13. ISOTROPIC CONCRETE
14. E 21.0
15. POISSON 0.17
16. DENSITY 2.36158E-008
17. ALPHA 5E-006
18. DAMP 0.05
19. G 9.25
20. TYPE CONCRETE
21. STRENGTH FCU 0.0275
22. END DEFINE MATERIAL
23. CONSTANTS
24. MATERIAL CONCRETE ALL
25. LOADING 1 PRESTRESSING LOAD
26. MEMBER PRESTRESS
27. 7 8 FORCE 1350. ES 75. EM -300. EE 75.
28. LOADING 2 POSTSTRESSING LOAD
29. MEMBER POSTSTRESS
30. 7 8 FORCE 1350. ES 75. EM -300. EE 75.
31. PERFORM ANALYSIS
32. FRAME WITH PRESTRESSING LOAD
33. ------ PROBLEM STATISTICS ------
34. NUMBER OF JOINTS 8  NUMBER OF MEMBERS 9
35. NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
36. NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 2
37. Using 64-bit analysis engine.
38. SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
39. TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 18
40. TOTAL LOAD COMBINATION CASES = 0 SO FAR.
41. UNIT METER
42. PRINT ANALYSIS RESULTS
43. ------ ANALYSIS RESULTS ------
44. FRAME WITH PRESTRESSING LOAD
45. ------ JOINT DISPLACEMENT (CM  RADIAN) ------
46. STRUCTURE TYPE = PLANE
47. ---------- JOINT LOAD ----------
48. LOAD X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN
49. 1 1
50. 2 0
51. 3 1 0.1917 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000
52. 4 1 -0.1917 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000
53. 5 1 0.1799 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000
54. 6 1 -0.1799 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000
### Support Reactions - Unit KNS ME TE

**Structure Type = Plane**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-30.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>80.49</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>30.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-80.49</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

### Member End Forces - Structure Type = Plane

**All Units Are -- KNS ME TE (Local)**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-0.00</td>
<td>30.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>80.49</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>-30.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>103.07</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>3</td>
<td>-0.00</td>
<td>62.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>121.89</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>-0.00</td>
<td>10.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>50.98</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>-10.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-4.64</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>0.00</td>
<td>-30.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-80.49</td>
</tr>
<tr>
<td>4</td>
<td>-0.00</td>
<td>30.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-103.07</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>4</td>
<td>0.00</td>
<td>-62.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-121.89</td>
</tr>
<tr>
<td>6</td>
<td>-0.00</td>
<td>62.59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-159.77</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>6</td>
<td>0.00</td>
<td>-10.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-50.98</td>
</tr>
<tr>
<td>8</td>
<td>-0.00</td>
<td>10.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>4.64</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>3</td>
<td>1371.41</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-326.21</td>
</tr>
<tr>
<td>4</td>
<td>-1371.41</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>326.21</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>1339.41</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-101.25</td>
</tr>
<tr>
<td>4</td>
<td>-1339.41</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>101.25</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>5</td>
<td>1287.12</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-312.00</td>
</tr>
<tr>
<td>6</td>
<td>-1287.12</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>312.00</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>1339.41</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-101.25</td>
</tr>
<tr>
<td>6</td>
<td>-1339.41</td>
<td>-168.75</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>101.25</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>7</td>
<td>-10.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>4.64</td>
</tr>
<tr>
<td>8</td>
<td>10.30</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-4.64</td>
</tr>
</tbody>
</table>
EX. UK-7 Modeling Offset Connections in a Frame

This example illustrates modeling of structures with offset connections. Offset connections arise when the center lines of the connected members do not intersect at the connection point. The connection eccentricity behaves as a rigid link and is modeled through specification of MEMBER OFFSETS.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-7 Modeling Offset Connections in a Frame.STD when you install the program.

Figure 489: Example Problem No. 7
Application Examples
EX. British Design Examples

Where:

\[
L_1 = 4.5 \text{ m} \\
L_2 = 6 \text{ m} \\
L_3 = 178 \text{ mm} \\
L_4 = 152 \text{ mm}
\]

Actual input is shown in bold lettering followed by explanation.

### STAAD PLANE TEST FOR MEMBER OFFSETS

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

### UNIT METER KNS

Defines the input units for the data that follows.

<table>
<thead>
<tr>
<th>JOINT COORD</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0. 0. ; 2 6. 0. ; 3 0. 4.5</td>
</tr>
<tr>
<td>4 6. 4.5 ; 5 0. 9. ; 6 6. 9.</td>
</tr>
</tbody>
</table>

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

### SUPPORT

| 1 2 PINNED |

Pinned supports are specified at joints 1 and 2. The word PINNED signifies that no moments will be carried by these supports.

### MEMB INCI

<table>
<thead>
<tr>
<th>MEMB INCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 3 2 ; 3 3 5 4</td>
</tr>
<tr>
<td>5 3 4 ; 6 5 6 ; 7 1 4</td>
</tr>
</tbody>
</table>

Defines the members by the joints to which they are connected.

### MEMB PROP BRITISH

<table>
<thead>
<tr>
<th>MEMB PROP BRITISH</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 4 TABLE ST UC356X368X129</td>
</tr>
<tr>
<td>5 6 TA ST UB305X165X40</td>
</tr>
<tr>
<td>7 TA LD UA200X150X12</td>
</tr>
</tbody>
</table>

All member properties are from British steel table. The word ST stands for standard single section. LD stands for long leg back-to-back double angle.

### UNIT MMS

### MEMB OFFSET

<table>
<thead>
<tr>
<th>MEMB OFFSET</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 6 START 178. 0.0 0.0</td>
</tr>
<tr>
<td>5 6 END -178. 0.0 0.0</td>
</tr>
<tr>
<td>7 END -178.0 -152.0 0.0</td>
</tr>
</tbody>
</table>

The preceding specification states that an OFFSET is located at the START/END joint of the members. The X, Y, and Z global coordinates of the offset distance from the corresponding incident joint are also provided. These attributes are applied to members 5, 6, and 7.

### DEFINE MATERIAL START

<table>
<thead>
<tr>
<th>DEFINE MATERIAL START</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td>E 210</td>
</tr>
<tr>
<td>POISSON 0.3</td>
</tr>
</tbody>
</table>
The **DEFINE MATERIAL** command is used to specify material properties and the **CONSTANT** is used to assign the material to all members.

**LOADING 1 WIND LOAD**

Load case 1 is initiated followed by a title.

**JOINT LOAD**

\[ \begin{align*}
3 & \text{ FX 225.0} \\
5 & \text{ FX 112.5}
\end{align*} \]

Load 1 contains joint loads at nodes 3 and 5. FX indicates that the load is a force in the global X direction.

**PERFORM ANALYSIS**

The above command is an instruction to perform the analysis.

**UNIT METER**

**PRINT FORCES**

**PRINT REACTIONS**

The above PRINT commands are instructions for writing the member forces and support reactions to the output file. The preceding line causes the results to be written in the length unit of meters.

**FINISH**

This command terminates a STAAD run.

**Input File**

**STAAD PLANE TEST FOR MEMBER OFFSETS**

**UNIT METER KNS**

**JOINT COORD**

\[ \begin{align*}
1 & \text{ 0. 0. ; 2 6. 0. ; 3 0. 4.5} \\
4 & \text{ 6. 4.5 ; 5 0. 9. ; 6 6. 9.}
\end{align*} \]

**SUPPORT**

1 2 PINNED

**MEMB INCI**

1 1 3 2 ; 3 3 5 4 \\
5 3 4 ; 6 5 6 ; 7 1 4

**MEMB PROP BRITISH**

1 TO 4 TABLE ST UC356X368X129 \\
5 6 TA ST UB305X165X40 \\
7 TA LD UA200X150X12

**UNIT MMS**

**MEMB OFFSET**

5 6 START 178. 0.0 0.0 \\
5 6 END -178. 0.0 0.0 \\
7 END -178.0 -152.0 0.0

**DEFINE MATERIAL START**

ISOTROPIC STEEL

\[ E \ 210 \]

**POISSON 0.3**

**DENSITY 7.6977e-008**
STAAD Output File

*************
*           STAAD.Pro CONNECT Edition             *
*           Version 22.01.00.**                    *
*           Proprietary Program of                 *
*           Bentley Systems, Inc.                  *
*           Date= APR 14, 2019                     *
*           Time= 22:54:50                         *
*           Licensed to: Bentley Systems Inc       *
*************

1. STAAD PLANE TEST FOR MEMBER OFFSETS

INPUT FILE: UK-7 Modeling Offset Connections in a Frame.STD

2. UNIT METER KNS
3. JOINT COORD
4. 1 0. 0. ; 2 6. 0. ; 3 0. 4.5
5. 4 6. 4.5 ; 5 0. 9. ; 6 6. 9.
6. SUPPORT
7. 1 2 PINNED
8. MEMB INCI
9. 1 1 3 2 ; 3 3 5 4
10. 5 3 4 ; 6 5 6 ; 7 1 4
11. MEMB PROP BRITISH
12. 1 T0 4 TABLE ST UC356X368X129
13. 5 6 TA ST UB305X165X40
14. 7 TA LD UA200X150X12
15. UNIT MMS
16. MEMB OFFSET
17. 5 6 START 178. 0.0 0.0
18. 5 6 END -178. 0.0 0.0
19. 7 END -178.0 -152.0 0.0
20. DEFINE MATERIAL START
21. ISOTROPIC STEEL
22. E 210
23. POISSON 0.3
24. DENSITY 7.6977E-008
25. ALPHA 6E-006
26. DAMP 0.03
27. TYPE STEEL
28. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
29. END DEFINE MATERIAL
30. CONSTANS
31. MATERIAL STEEL ALL
32. LOADING 1 WIND LOAD
33. JOINT LOAD
34. 3 FX 225.0 ; 5 FX 112.5
35. PERFORM ANALYSIS

TEST FOR MEMBER OFFSETS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS       7
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
36. UNIT METER
37. PRINT FORCES

FORCES

TEST FOR MEMBER OFFSETS
MEMBER END FORCES STRUCTURE TYPE = PLANE

------------
ALL UNITS ARE -- KNS METE (LOCAL )
MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
1 1 1 -49.14 20.23 0.00 0.00 0.00 4.59
3 1 49.14 20.23 0.00 0.00 0.00 95.65
2 1 2 337.50 25.11 0.00 0.00 0.00 0.00
4 1 3 337.50 25.11 0.00 0.00 0.00 -112.99
3 1 3 -30.77 53.79 0.00 0.00 0.00 150.98
5 1 5 -30.77 53.79 0.00 0.00 0.00 91.06
4 1 4 30.77 58.71 0.00 0.00 0.00 170.67
6 1 6 -30.77 58.71 0.00 0.00 0.00 93.55
5 1 3 299.02 -18.38 0.00 0.00 0.00 -52.06
4 1 5 299.02 -18.38 0.00 0.00 0.00 -51.65
6 1 5 58.71 -30.77 0.00 0.00 0.00 -85.58
6 1 6 -58.71 30.77 0.00 0.00 0.00 -88.07
7 1 1 479.29 -1.96 0.00 0.00 0.00 -4.59
4 1 4 479.29 1.96 0.00 0.00 0.00 -9.62

************** END OF LATEST ANALYSIS RESULT **************

38. PRINT REACTIONS

REACTION

TEST FOR MEMBER OFFSETS
SUPPORT REACTIONS -UNIT KNS METE STRUCTURE TYPE = PLANE

------------
JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM-Z
1 1 -362.61 -337.50 0.00 0.00 0.00 0.00
2 1  25.11 337.50 0.00 0.00 0.00 0.00

************** END OF LATEST ANALYSIS RESULT **************

39. FINISH

*********** END OF THE STAAD.Pro RUN ***********
**** DATE= APR 14,2019  TIME= 22:54:51 ****

TEST FOR MEMBER OFFSETS

**********************************************************************
* For technical assistance on STAAD.Pro, please visit *
*
EX. UK-8 Concrete Design for a Space Frame

In this example, concrete design is performed on some members of a space frame structure. Design calculations consist of computation of reinforcement for beams and columns. Secondary moments on the columns are obtained through the means of a P-Delta analysis.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-8 Concrete Design for a Space Frame.STD when you install the program.

![Figure 490: Example Problem No. 8](image)

The above example represents a space frame, and the members are made of concrete. The input in the next page will show the dimensions of the members.

Two load cases, namely one for dead plus live load and another with dead, live and wind load, are considered in the design.
STAAD SPACE FRAME WITH CONCRETE DESIGN

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

UNIT METER KNS

Defines the input units for the data that follows.

JOINT COORDINATE
1 0 0 0 ; 2 5.4 0 0 ; 3 11.4 0 0
4 0 0 7.2 ; 5 5.4 0 7.2 ; 6 11.4 0 7.2
7 0 3.6 0 ; 8 5.4 3.6 0 ; 9 11.4 3.6 0
10 0 3.6 7.2 ; 11 5.4 3.6 7.2 ; 12 11.4 3.6 7.2
13 5.4 7.2 0 ; 14 11.4 7.2 0 ; 15 5.4 7.2 7.2
16 11.4 7.2 7.2

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCE
1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
9 6 12 ; 10 12 16 ; 11 7 8 12
13 10 11 14 ; 15 13 14 ; 16 15 16
17 7 10 ; 18 8 11 ; 19 9 12
20 13 15 ; 21 14 16

Defines the members by the joints to which they are connected.

UNIT MMS
MEMB PROP
1 2 PRISMATIC YD 300.0 IZ 2.119E08 IY 2.119E08 –
IX 4.237E08
3 TO 10 PR YD 300.0 ZD 300.0 IZ 3.596E08 IY 3.596E08 –
IX 5.324E08
11 TO 21 PR YD 535.0 ZD 380 IZ 2.409E09 IY 1.229E09 –
IX 2.704E09

All member properties are provided using the PRISMATIC option. YD and ZD stand for depth and width. If ZD is not provided, a circular shape with diameter = YD is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. For this particular example, moments of inertia (IZ, IY) and torsional constant (IX) are provided, so these will not be recalculated. The IX, IY, and IZ values provided in this example are only half the values of a full section to account for the fact that the full moments of inertia will not be effective due to cracking of concrete.

DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.356e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
The DEFINE MATERIAL command is used to define material properties for concrete. The CONSTANT command is used to assign this to all members.

SUPPORT
1 TO 6 FIXED

Joints 1 to 6 are fixed supports.

LOAD 1 (1.4DL + 1.7LL)

Load case 1 is initiated followed by a title.

SELF Y -1.4

The selfweight of the structure is applied in the global Y direction with a -1.4 factor. Since global Y is vertically upward, the negative factor indicates that this load will act downwards.

MEMB LOAD
11 TO 16 UNI Y -42.0
11 TO 16 UNI Y -76.5

Load 1 contains member loads also. Y indicates that the load is in the local Y direction. The word UNI stands for uniformly distributed load.

LOAD 2 .75 (1.4DL + 1.7LL + 1.7WL)

Load case 2 is initiated followed by a title.

REPEAT LOAD
1 0.75

The preceding command will gather the load data values from load case 1, multiply them with a factor of 0.75 and utilize the resulting values in load 2.

JOINT LOAD
15 16 FZ 40.0
11 FZ 90.0
12 FZ 70.0
10 FZ 40.0

Load 2 contains some additional joint loads also. FZ indicates that the load is a force in the global Z direction.

PDELTA ANALYSIS

This command instructs the program to proceed with the analysis. The analysis type is P-DELTA indicating that second-order effects are to be calculated.

PRINT FORCES LIST 2 5 9 14 16

Member end forces are printed using the above PRINT commands. The LIST option restricts the print output to the members listed.

START CONCRETE DESIGN

The above command initiates a concrete design.

CODE BRITISH
TRACK 1.0 MEMB 14
TRACK 2.0 MEMB 16
MAXMAIN 40 ALL
The values for the concrete design parameters are defined in the above commands. Design is performed per the BS 8110 Code. The TRACK value dictates the extent of design related information provided in the output. MAXMAIN indicates that the maximum size of main reinforcement is the 40 mm bar. These parameters are described in the manual where British concrete design related information is available.

<table>
<thead>
<tr>
<th>COMMAND</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>DESIGN BEAM 14 16</td>
<td>The above command instructs the program to design beams 14 and 16 for flexure, shear, and torsion.</td>
</tr>
<tr>
<td>DESIGN COLUMN 2 5</td>
<td>The above command instructs the program to design columns 2 and 5 for axial load and biaxial bending.</td>
</tr>
</tbody>
</table>

This will end the concrete design.

<table>
<thead>
<tr>
<th>COMMAND</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>FINISH</td>
<td>This command terminates the STAAD run.</td>
</tr>
</tbody>
</table>

### Input File

```
STAAD SPACE FRAME WITH CONCRETE DESIGN
UNIT METER KNS
JOINT COORDINATE
  1  0.0  0.0  0.0 ;  2  5.4  0.0  0.0
  3 11.4  0.0  0.0 ;  4  0.0  0.0  7.2
  5  5.4  0.0  7.2 ;  6 11.4  0.0  7.2
  7  0.0  3.6  0.0 ;  8  5.4  3.6  0.0
  9 11.4  3.6  0.0 ; 10  0.0  3.6  7.2
 11  5.4  3.6  7.2 ; 12 11.4  3.6  7.2
 13  5.4  7.2  0.0 ; 14 11.4  7.2  0.0
 15  5.4  7.2  7.2 ; 16 11.4  7.2  7.2
MEMBER INCIDENCE
  1  1  7 ;  2  4 10 ;  3  2  8 ;  4  8 13
  5  5 11 ;  6 11 15 ;  7  3  9 ;  8  9 14
  9  6 12 ; 10 12 16 ; 11  7  8 12
 13 10 11 14 ; 15 13 14 ; 16 15 16
 17  7 10 ; 18  8 11 ; 19  9 12
 20 13 15 ; 21 14 16
UNIT MMS
MEMB PROP
  1  2  PRISMATIC YD 300.0 ZD 300.0 IZ 2.119E08 IY 2.119E08 -
  3 TO 10 PR YD 300.0 ZD 300.0 IZ 3.596E08 IY 3.596E08 -
           IX 5.324E08
 11 TO 21 PR YD 535.0 ZD 380 IZ 2.409E09 IY 1.229E09 -
           IX 2.704E09
DEFINE MATERIAL START
  ISOTROPIC CONCRETE
  E 21.0
  POISSON 0.17
  DENSITY 2.356e-008
  ALPHA 5e-006
  DAMP 0.05
  G 9.25
  TYPE CONCRETE
  STRENGTH FCU 0.0275
END DEFINE MATERIAL
```
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
SUPPORT
1 TO 6 FIXED
LOAD 1 (1.4DL + 1.7LL)
SELF Y -1.4
MEMB LOAD
11 TO 16 UNI Y -42.0
11 TO 16 UNI Y -76.5
LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
REPEAT LOAD
1 0.75
JOIN LOAD
15 16 FZ 40.0
11 FZ 90.0
12 FZ 70.0
10 FZ 40.0
PDELTA ANALYSIS
PRINT FORCES LIST 2 5 9 14 16
START CONCRETE DESIGN
CODE BS8007
TRACK 1.0 MEMB 14
TRACK 2.0 MEMB 16
MAXMAIN 40 ALL
DESIGN BEAM 14 16
DESIGN COLUMN 2 5
END CONCRETE DESIGN
FINISH

1. STAAD SPACE FRAME WITH CONCRETE DESIGN
INPUT FILE: UK-8 Concrete Design for a Space Frame.STD
2. UNIT METER KNS
3. JOINT COORDINATE
4. 1 0.0 0.0 0.0 ; 2 5.4 0.0 0.0
5. 3 11.4 0.0 0.0 ; 4 0.0 0.0 7.2
6. 5 5.4 0.0 7.2 ; 6 11.4 0.0 7.2
7. 7 0.0 3.6 0.0 ; 8 5.4 3.6 0.0
8. 9 11.4 3.6 0.0 ; 10 0.0 3.6 7.2
9. 11 5.4 3.6 7.2 ; 12 11.4 3.6 7.2
10. 13 5.4 7.2 0.0 ; 14 11.4 7.2 0.0
11. 15 5.4 7.2 7.2 ; 16 11.4 7.2 7.2
12. MEMBER INCIDENCE
13. 1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
14. 5 5 11; 6 11 15; 7 3 9; 8 9 14
15. 9 6 12; 10 12 16; 11 7 8 12
16. 13 10 11 14; 15 13 14; 16 15 16
17. 17 7 10; 18 8 11; 19 9 12
18. 20 13 15; 21 14 16
19. UNIT MMS
20. MEMB PROP
21. 1 2 PRISMATIC YD 300.0 IZ 2.119E08 IY 2.119E08 -
22. IX 4.237E08
23. 3 TO 10 PR YD 300.0 ZD 300.0 IZ 3.596E08 IY 3.596E08 -
24. IX 5.324E08
25. 11 TO 21 PR YD 535.0 ZD 380 IZ 2.409E09 IY 1.229E09 -
26. IX 2.704E09
27. DEFINE MATERIAL START
28. ISOTROPIC CONCRETE
29. E 21.0
30. POISSON 0.17
31. DENSITY 2.356E-008
32. ALPHA 5E-006
33. DAMP 0.05
34. G 9.25
35. TYPE CONCRETE
36. STRENGTH FCU 0.0275
37. END DEFINE MATERIAL
38. CONSTANTS
39. FRAME WITH CONCRETE DESIGN -- PAGE NO. 2
40. MATERIAL CONCRETE ALL
41. UNIT METER
42. SUPPORT
43. 1 TO 6 FIXED
44. SELF Y -1.4
45. MEMB LOAD
46. 11 TO 16 UNI Y -42.0
47. 11 TO 16 UNI Y -76.5
48. LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
49. REPEAT LOAD
50. 1 0.75
51. JOINT LOAD
52. 15 16 FZ 40.0
53. 11 FZ 90.0
54. 12 FZ 70.0
55. 10 FZ 40.0
56. PDELTA ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS 16  NUMBER OF MEMBERS 21
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 6

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 60
TOTAL LOAD COMBINATION CASES = 0 SO FAR.
++ Adjusting Displacements.
57. PRINT FORCES LIST 2 5 9 14 16
58. FORCES LIST 2 5
59. FRAME WITH CONCRETE DESIGN -- PAGE NO. 3
60. MEMBER END FORCES STRUCTURE TYPE = SPACE
---

**ALL UNITS ARE -- KNS METE (LOCAL)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
<td>4</td>
<td>301.86</td>
<td>-18.04</td>
<td>-2.84</td>
<td>0.00</td>
<td>3.49</td>
<td>-23.83</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td></td>
<td>-293.46</td>
<td>18.04</td>
<td>2.84</td>
<td>0.00</td>
<td>6.73</td>
<td>-41.68</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>4</td>
<td>243.90</td>
<td>-14.96</td>
<td>-30.23</td>
<td>1.39</td>
<td>57.98</td>
<td>-20.50</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td></td>
<td>-237.60</td>
<td>14.96</td>
<td>30.23</td>
<td>-1.39</td>
<td>57.04</td>
<td>-33.98</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>5</td>
<td>1290.04</td>
<td>-2.02</td>
<td>-2.94</td>
<td>0.00</td>
<td>3.79</td>
<td>-5.67</td>
</tr>
<tr>
<td></td>
<td>11</td>
<td></td>
<td>-1279.35</td>
<td>2.02</td>
<td>2.94</td>
<td>0.00</td>
<td>6.82</td>
<td>-4.10</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>5</td>
<td>1015.94</td>
<td>-3.81</td>
<td>-55.16</td>
<td>1.18</td>
<td>121.20</td>
<td>-8.83</td>
</tr>
<tr>
<td></td>
<td>11</td>
<td></td>
<td>-1007.92</td>
<td>3.81</td>
<td>55.16</td>
<td>-1.18</td>
<td>111.88</td>
<td>-7.55</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>6</td>
<td>758.52</td>
<td>20.05</td>
<td>-2.75</td>
<td>0.00</td>
<td>3.44</td>
<td>22.20</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td></td>
<td>-747.84</td>
<td>20.05</td>
<td>2.75</td>
<td>0.00</td>
<td>6.48</td>
<td>48.55</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>6</td>
<td>620.14</td>
<td>12.89</td>
<td>-60.00</td>
<td>0.16</td>
<td>124.29</td>
<td>12.18</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td></td>
<td>-612.12</td>
<td>12.89</td>
<td>60.00</td>
<td>-0.16</td>
<td>113.38</td>
<td>32.62</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>11</td>
<td>-40.76</td>
<td>435.24</td>
<td>0.00</td>
<td>-0.29</td>
<td>0.01</td>
<td>498.88</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td></td>
<td>40.76</td>
<td>315.99</td>
<td>0.00</td>
<td>0.29</td>
<td>0.00</td>
<td>-141.08</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>11</td>
<td>-34.70</td>
<td>327.08</td>
<td>3.02</td>
<td>-1.14</td>
<td>-12.30</td>
<td>375.74</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td></td>
<td>34.70</td>
<td>236.34</td>
<td>-3.02</td>
<td>1.14</td>
<td>-5.89</td>
<td>-103.49</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>15</td>
<td>60.81</td>
<td>378.36</td>
<td>0.00</td>
<td>0.04</td>
<td>0.00</td>
<td>139.84</td>
</tr>
<tr>
<td></td>
<td>16</td>
<td></td>
<td>-60.81</td>
<td>372.88</td>
<td>0.00</td>
<td>-0.04</td>
<td>0.00</td>
<td>-123.47</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>15</td>
<td>45.61</td>
<td>283.60</td>
<td>0.40</td>
<td>-0.13</td>
<td>-1.29</td>
<td>104.67</td>
</tr>
<tr>
<td></td>
<td>16</td>
<td></td>
<td>-45.61</td>
<td>279.83</td>
<td>0.40</td>
<td>0.13</td>
<td>-1.01</td>
<td>-93.39</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

58. START CONCRETE DESIGN

CONCRETE DESIGN

59. CODE BS8007

PROGRAM CODE REVISION V1.0_8007_87/1

60. TRACK 1.0 MEMB 14

61. TRACK 2.0 MEMB 16

62. MAXMAIN 40 ALL

63. DESIGN BEAM 14 16

FRAME WITH CONCRETE DESIGN

---

**BE Am N O. 14 DE S I GN R ES U LT S - F LEXURE**

LEN - 6000. mm FY - 460. FC - 30. SIZE - 380. X 535. mm

LEVEL HEIGHT BAR INFO FROM TO ANCHOR

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>35</td>
<td>5- 20 MM 740. 6000. NO YES</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

CRITICAL POS MOMENT= 257.51 KN-M AT 3500. mm, LOAD 1
REQD STEEL= 1480. mm2, ROW=0.0073, ROWMX=0.0400, ROWMN=0.0013
MAX/MIN/ACTUAL BAR SPACING= 174./ 45./ 58. mm

---

2 487. 4- 32 MM 0. 1760. YES NO

COMP 5- 10 MM (REQD. STEEL= 375. SQ. MM)

---

CRITICAL NEG MOMENT= 498.88 KN-M AT 0. mm, LOAD 1
REQD STEEL= 3192. mm2, ROW=0.0157, ROWMX=0.0400, ROWMN=0.0013
MAX/MIN/ACTUAL BAR SPACING= 171./ 35./ 70. mm

---

3 497. 7- 12 MM 5240. 6000. NO YES

CRITICAL NEG MOMENT= 141.08 KN-M AT 6000. mm, LOAD 1
REQD STEEL= 762. mm2, ROW=0.0037, ROWMX=0.0400, ROWMN=0.0013
MAX/MIN/ACTUAL BAR SPACING= 170./ 37./ 41. mm

---
### PROVIDE SHEAR LINKS AS FOLLOWS

<table>
<thead>
<tr>
<th>FROM</th>
<th>TO</th>
<th>MAX. SHEAR</th>
<th>LOAD</th>
<th>LINKS</th>
<th>NO.</th>
<th>SPACING C/C</th>
</tr>
</thead>
<tbody>
<tr>
<td>END 1</td>
<td>2001 mm</td>
<td>435.2 kN</td>
<td>1</td>
<td>8 mm</td>
<td>28</td>
<td>74 mm</td>
</tr>
<tr>
<td>2001</td>
<td>3502 mm</td>
<td>184.8 kN</td>
<td>1</td>
<td>8 mm</td>
<td>6</td>
<td>250 mm</td>
</tr>
<tr>
<td>3499</td>
<td>5000 mm</td>
<td>190.8 kN</td>
<td>1</td>
<td>8 mm</td>
<td>6</td>
<td>250 mm</td>
</tr>
<tr>
<td>5000</td>
<td>END 2</td>
<td>316.0 kN</td>
<td>1</td>
<td>8 mm</td>
<td>10</td>
<td>111 mm</td>
</tr>
</tbody>
</table>

---

### FRAME WITH CONCRETE DESIGN

---

### EX. British Design Examples

---

### B E A M N O. 14 D E S I G N R E S U L T S - FLEXURE

**LEN** - 6000. mm  **FY** - 460.  **FC** - 30.  **SIZE** - 380. X 535. mm

<table>
<thead>
<tr>
<th>LEVEL</th>
<th>HEIGHT</th>
<th>BAR INFO</th>
<th>FROM</th>
<th>TO</th>
<th>ANCHOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>41.</td>
<td>4- 32 MM</td>
<td>0.</td>
<td>6000.</td>
<td>YES</td>
</tr>
<tr>
<td>2</td>
<td>497.</td>
<td>7- 12 MM</td>
<td>0.</td>
<td>760.</td>
<td>YES</td>
</tr>
<tr>
<td>3</td>
<td>497.</td>
<td>6- 12 MM</td>
<td>5240.</td>
<td>6000.</td>
<td>NO</td>
</tr>
</tbody>
</table>

**REQUIRED REINF. STEEL SUMMARY:**

<table>
<thead>
<tr>
<th>SECTION</th>
<th>REINF STEEL(+VE/-VE)</th>
<th>MOMENTS(+VE/-VE)</th>
<th>LOAD(+VE/-VE)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.</td>
<td>0.0/ 754.7</td>
<td>0.00/ 139.84</td>
<td>0/ 1</td>
</tr>
<tr>
<td>500.</td>
<td>264.3/ 0.0</td>
<td>34.13/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>1000.</td>
<td>971.9/ 0.0</td>
<td>176.79/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>1500.</td>
<td>1687.4/ 0.0</td>
<td>288.09/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>2000.</td>
<td>2278.7/ 0.0</td>
<td>368.01/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>2500.</td>
<td>2684.2/ 0.0</td>
<td>416.52/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>3000.</td>
<td>2829.9/ 264.3</td>
<td>433.60/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>3500.</td>
<td>2708.4/ 0.0</td>
<td>419.25/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>4000.</td>
<td>2322.3/ 0.0</td>
<td>373.48/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>4500.</td>
<td>1744.6/ 0.0</td>
<td>296.29/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>5000.</td>
<td>1037.9/ 0.0</td>
<td>187.72/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>5500.</td>
<td>264.3/ 0.0</td>
<td>47.78/ 0.0</td>
<td>1/ 0</td>
</tr>
<tr>
<td>6000.</td>
<td>0.0/ 661.2</td>
<td>0.00/ 123.47</td>
<td>0/ 1</td>
</tr>
</tbody>
</table>

---

### FRAME WITH CONCRETE DESIGN
<table>
<thead>
<tr>
<th>FROM</th>
<th>TO</th>
<th>MAX. SHEAR</th>
<th>LOAD</th>
<th>LINKS</th>
<th>NO.</th>
<th>SPACING C/C</th>
</tr>
</thead>
<tbody>
<tr>
<td>END 1 2001 mm</td>
<td>378.4 kN</td>
<td>1</td>
<td>8 mm</td>
<td>28</td>
<td>74 mm</td>
<td></td>
</tr>
<tr>
<td>2001 3002 mm</td>
<td>127.9 kN</td>
<td>1</td>
<td>8 mm</td>
<td>4</td>
<td>250 mm</td>
<td></td>
</tr>
<tr>
<td>2999 4000 mm</td>
<td>122.5 kN</td>
<td>1</td>
<td>8 mm</td>
<td>4</td>
<td>250 mm</td>
<td></td>
</tr>
<tr>
<td>4000 END 2</td>
<td>372.9 kN</td>
<td>1</td>
<td>8 mm</td>
<td>28</td>
<td>74 mm</td>
<td></td>
</tr>
</tbody>
</table>

---

For technical assistance on STAAD.Pro, please visit http://www.bentley.com/en/support/
Details about additional assistance from Bentley and Partners can be found at program menu.
EX. UK-9 Modeling Slabs and Shear Walls Using Finite Elements

The space frame structure in this example consists of frame members and finite elements (plates). The finite element part is used to model floor slabs and a shear wall. Concrete design of an element is performed. This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-9 Modeling Slabs and Shear Walls Using Finite Elements.STD when you install the program.

Actual input is shown in bold lettering followed by explanation.

STAAD SPACE
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND FINITE ELEMENTS

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The second line forms the title to identify this project.
The units for the data that follows are specified above.

```
JOINT COORD
  1 0 0 0 ; 2 0 0 6
  7 0 4.5 0 11 0 4.5 6
  12 1.5 4.5 0 14 4.5 4.5 0
  18 6 4.5 0 22 6 4.5 6
  23 7.5 4.5 0 25 10.5 4.5 0
  26 7.5 4.5 6 28 10.5 4.5 6
  29 12 4.5 0 33 12 4.5 6
  34 6 1.125 0 36 6 3.375 0
  37 6 1.125 6 39 6 3.375 6
```

The joint numbers and their coordinates are defined through the above set of commands. The automatic generation facility has been used several times in the above lines. See section 5.1.1 of the Technical Reference Manual (on page 2425) where the joint coordinate generation facilities are described.

```
MEMBER INCI
  *COLUMNS
    1 1 7 ; 2 2 11
    3 3 34 ; 4 34 35 ; 6 35 36 ; 10 36 18
    7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
    11 5 29 ; 12 6 33
  *BEAMS IN Z DIRECTION AT X=0
    13 7 8 16
  *BEAMS IN Z DIRECTION AT X=6.0
    17 18 19 20
  *BEAMS IN Z DIRECTION AT X=12.0
    21 29 30 24
  *BEAMS IN X DIRECTION AT Z = 0
    25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
    29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
  *BEAMS IN X DIRECTION AT Z = 12.0
    33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
    37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
```

The member incidences are defined through the above set of commands. For some members, the member number followed by the start and end joint numbers are defined. In other cases, STAAD's automatic generation facilities are utilized. Section 5 of the Technical Reference Manual describes these facilities in detail.

```
DEFINE MESH
  A JOINT 7
  B JOINT 11
  C JOINT 22
  D JOINT 18
  E JOINT 33
  F JOINT 29
  G JOINT 3
  H JOINT 4
```

The above lines define the nodes of super-elements. Super-elements are plate/shell surfaces from which a number of individual plate/shell elements can be generated. In this case, the points describe the outer corners of a slab and that of a shear wall. Our goal is to define the slab and the wall as several plate/shell elements.

```
GENERATE ELEMENT
  MESH ABCD 4 4
  MESH DCEF 4 4
  MESH DCHG 4 4
```
The above lines form the instructions to generate individual 4-noded elements from the super-element profiles. For example, the command MESH ABCD 4 4 means that STAAD.Pro has to generate 16 elements from the surface formed by the points A, B, C and D with 4 elements along the edges AB & CD and 4 elements along the edges BC & DA.

```
UNIT MMS
MEMB PROP
  1 TO 40 PRIS YD 300 ZD 300
```

Members 1 to 40 are defined as a rectangular prismatic section with 300 mm depth and 300 mm width.

```
ELEM PROP
  41 TO 88 TH 150
```

Elements 41 to 88 are defined to be 150 mm thick.

```
DEFINE MATERIAL START
  ISOTROPIC CONCRETE
  E 21.0
  POISSON 0.17
  DENSITY 2.36158e-008
  ALPHA 5e-006
  DAMP 0.05
  G 9.25
  TYPE CONCRETE
  STRENGTH FCU 0.0275
  END DEFINE MATERIAL
  CONSTANTS
  MATERIAL CONCRETE ALL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
SUPPORT
  1 TO 6 FIXED
```

Joints 1 to 6 are defined as fixed supported.

```
UNIT KNS METER
LOAD 1 DEAD LOAD FROM FLOOR
  ELEMENT LOAD
    41 TO 72 PRESSURE -10.0
LOAD 2 WIND LOAD
  JOINT LOAD
    11 33 FZ -90.
    22 FZ -450.
```

Load 1 consists of a pressure load of 10 KNS/sq.m. The intensity on elements 41 to 72. The negative sign (and the default value for the axis) indicates that the load acts opposite to the positive direction of the element local z-axis.

```
LOAD 2 WIND LOAD
  JOINT LOAD
    11 33 FZ -90.
    22 FZ -450.
```

Load 2 consists of joint loads in the Z direction at joints 11, 22, and 33.

```
LOAD COMB 3
  1 0.9 2 1.3
```

Load 3 is a combination of 0.9 times load case 1 and 1.3 times load case 2.

```
PERFORM ANALYSIS
```
The command to perform an elastic analysis is specified above.

```
LOAD LIST 1 3
PRINT SUPP REAC
PRINT MEMBER FORCES LIST 27
PRINT ELEMENT STRESSES LIST 47
```

Support reactions, members forces and element stresses are printed for load cases 1 and 3.

```
START CONCRETE DESIGN
CODE BRITISH
DESIGN ELEMENT 47
END CONCRETE DESIGN
```

The above set of command form the instructions to STAAD to perform a concrete design on element 47. Design is done according to the British code. Note that design will consist only of flexural reinforcement calculations in the longitudinal and transverse directions of the elements for the moments MX and MY.

```
FINI
```

The STAAD run is terminated.

**Input File**

```
STAAD SPACE
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
* FINITE ELEMENTS
UNIT METER NEWTON
JOINT COORD
  1 0 0 0 ; 2 0 0 6.0
REP ALL 2 6.0 0 0
  7 0 4.5 0 11 0 4.5 6.0
  12 1.5 4.5 0 14 4.5 4.5 0
  15 1.5 4.5 6.0 17 4.5 4.5 6.0
  18 6.0 4.5 0 22 6.0 4.5 6.0
  23 7.5 4.5 0 25 10.5 4.5 0
  26 7.5 4.5 6.0 28 10.5 4.5 6.0
  29 12. 4.5 0 33 12. 4.5 6.0
  34 6.0 1.125 0 36 6.0 3.375 0
  37 6.0 1.125 6.0 39 6.0 3.375 6.0
MEMBER INCI
*COLUMNS
  1 1 7 ; 2 2 11
  3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
  7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
  11 5 29 ; 12 6 33
*BEAMS IN Z DIRECTION AT X=0
  13 7 8 16
*BEAMS IN Z DIRECTION AT X=6.0
  17 18 19 20
*BEAMS IN Z DIRECTION AT X=12.0
  21 29 30 24
*BEAMS IN X DIRECTION AT Z = 0
  25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
  29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
*BEAMS IN X DIRECTION AT Z = 12.0
  33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
  37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
DEFINE MESH
A JOINT 7
```
B JOINT 11
C JOINT 22
D JOINT 18
E JOINT 33
F JOINT 29
G JOINT 3
H JOINT 4
GENERATE ELEMENT
MESH ABCD 4 4
MESH DCEF 4 4
MESH DCHG 4 4
UNIT MMS
MEMB PROP
1 TO 40 PRIS YD 300 ZD 300
ELEM PROP
41 TO 88 TH 150
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 6 FIXED
UNIT METER
LOAD 1 DEAD LOAD FROM FLOOR
ELEMENT LOAD
41 TO 72 PRESSURE -10.0
LOAD 2 WIND LOAD
JOINT LOAD
11 33 FZ -90.
22 FZ -450.
LOAD COMB 3
1 0.9 2 1.3
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT SUPP REAC
PRINT MEMBER FORCES LIST 27
PRINT ELEMENT STRESSES LIST 47
START CONCRETE DESIGN
CODE BS8007
DESIGN ELEMENT 47
END CONCRETE DESIGN
FINI

**Application Examples**

**EX. British Design Examples**

---

**STAAD Output File**
1. STAAD SPACE
INPUT FILE: UK-9 Modeling Slabs and Shear Walls Using Finite Elements.STD
2. * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
3. * FINITE ELEMENTS
4. UNIT METER NEWTON
5. JOINT COORD
6. 1 0 0 0 ; 2 0 0 6.0
7. REP ALL 2 6.0 0 0
8. 7 0 4.5 0 11 0 4.5 6.0
9. 12 1.5 4.5 0 14 4.5 4.5 0
10. 15 1.5 4.5 6.0 17 4.5 4.5 6.0
11. 18 6.0 4.5 0 22 6.0 4.5 6.0
12. 23 7.5 4.5 0 25 10.5 4.5 0
13. 26 7.5 4.5 6.0 28 10.5 4.5 6.0
14. 29 12. 4.5 0 33 12. 4.5 6.0
15. 34 6.0 1.125 0 36 6.0 3.375 0
16. 37 6.0 1.125 6.0 39 6.0 3.375 6.0
17. MEMBER INCI
18. *COLUMNS
19. 1 1 7 ; 2 2 11
20. 3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
21. 7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
22. 11 5 29 ; 12 6 33
23. *BEAMS IN Z DIRECTION AT X=0
24. 13 7 8 16
25. *BEAMS IN Z DIRECTION AT X=6.0
26. 17 18 19 20
27. *BEAMS IN Z DIRECTION AT X=12.0
28. 21 29 30 24
29. *BEAMS IN X DIRECTION AT Z = 0
30. 25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
31. 29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
32. *BEAMS IN X DIRECTION AT Z = 12.0
33. 33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
34. 37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
35. DEFINE MESH
36. A JOINT 7
37. B JOINT 11
38. C JOINT 22
39. D JOINT 18
40. E JOINT 33
41. F JOINT 29
42. G JOINT 3
43. H JOINT 4
44. GENERATE ELEMENT
45. MESH ABCD 4 4
46. MESH DCEF 4 4
47. MESH DCHG 4 4
48. UNIT MMS
49. MEMB PROP
50. 1 TO 40 PRIS YD 300 ZD 300
51. ELEM PROP
52. 41 TO 88 TH 150
53. UNIT KNS MMS
54. DEFINE MATERIAL START
55. ISOTROPIC CONCRETE
56. E 21.0
57. POISSON 0.17
58. DENSITY 2.36158E-008
59. ALPHA 5E-006
60. DAMP 0.05
61. G 9.25
62. TYPE CONCRETE
63. STRENGTH FCU 0.0275
64. END DEFINE MATERIAL
65. CONSTANTS
66. MATERIAL CONCRETE ALL
67. SUPPORT
68. 1 TO 6 FIXED
69. UNIT METER
70. LOAD 1 DEAD LOAD FROM FLOOR
71. ELEMENT LOAD
72. 41 TO 72 PRESSURE -10.0
73. LOAD 2 WIND LOAD
74. JOINT LOAD
75. 11 33 FZ -90.
76. 22 FZ -450.
77. LOAD COMB 3
78. 1 0.9 2 1.3
79. PERFORM ANALYSIS

**PROBLEM STATISTICS**

| NUMBER OF JOINTS | 69 | NUMBER OF MEMBERS | 40 |
| NUMBER OF PLATES | 48 | NUMBER OF SOLIDS   | 0  |
| NUMBER OF SURFACES| 0  | NUMBER OF SUPPORTS | 6  |

Using 64-bit analysis engine.

* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
  SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER

TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 378
TOTAL LOAD COMBINATION CASES = 1 SO FAR.

80. LOAD LIST 1 3
81. PRINT SUPP REAC

**SUPP REAC**

* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
  SUPPORT REACTIONS -UNIT KNS METE STRUCTURE TYPE = SPACE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>8.20</td>
<td>74.30</td>
<td>10.34</td>
<td>15.41</td>
<td>-0.00</td>
<td>-12.21</td>
</tr>
<tr>
<td>3</td>
<td>7.48</td>
<td>68.32</td>
<td>10.82</td>
<td>17.66</td>
<td>0.05</td>
<td>-11.06</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>8.20</td>
<td>74.30</td>
<td>-10.34</td>
<td>-15.41</td>
<td>0.00</td>
<td>-12.21</td>
</tr>
<tr>
<td>3</td>
<td>7.30</td>
<td>65.43</td>
<td>-7.74</td>
<td>-9.89</td>
<td>0.24</td>
<td>-11.01</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.00</td>
<td>211.41</td>
<td>68.03</td>
<td>-10.57</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.00</td>
<td>791.81</td>
<td>476.42</td>
<td>12.46</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>
4 1 -0.00 211.41 -68.03 10.57 0.00 0.00
3 -0.00 -411.31 336.42 30.82 0.00 12.21
5 1 -8.20 74.30 10.34 15.41 0.00 0.00
3 -7.48 68.32 10.82 17.66 -0.05 11.06
6 1 -8.20 74.30 -10.34 -15.41 -0.00 12.21
3 -7.30 65.43 -7.74 -9.89 -0.24 11.01

************** END OF LATEST ANALYSIS RESULT **************

82. PRINT MEMBER FORCES LIST 27
MEMBER FORCES LIST 27
STAAD SPACE -- PAGE NO. 5
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
MEMBER END FORCES  STRUCTURE TYPE = SPACE
-------------------
ALL UNITS ARE -- KNS  METE (LOCAL )

MEMBER LOAD JT AXIAL SHEAR-Y SHEAR-Z TORSION MOM-Y MOM-Z
27 1 13 0.66 -11.91 -0.06 6.91 0.05 -21.61
14 -0.66 11.91 0.06 -6.91 0.05 3.75
3 13 22.07 -10.55 -0.71 6.42 0.52 -19.50
14 -22.07 10.55 0.71 -6.42 0.54 3.68

************** END OF LATEST ANALYSIS RESULT **************

83. PRINT ELEMENT STRESSES LIST 47
ELEMENT STRESSES LIST 47
STAAD SPACE -- PAGE NO. 6
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
ELEMENT STRESSES  FORCE,LENGTH UNITS= KNS  METE
-------------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

ELEMENT LOAD SQX SQY MX MY MXY VONT VONB SX SY SXY TREScatid TREScab
47 1 17.12 4.80 -10.40 -13.32 1.30
3305.41 3272.53 -12.02 -16.95 5.17
3704.34 3664.81
TOP: SMAX= -2648.50 SMIN= -3704.34 TMAX= 527.92 ANGLE= 21.0
BOTT: SMAX= 3664.81 SMIN= 2630.10 TMAX= 517.36 ANGLE= -69.2
3 14.94 4.42 -9.50 -11.97 1.01
3075.62 2867.90 -47.26 -61.01 180.98
3479.94 3143.47
TOP: SMAX= -2353.91 SMIN= -3479.94 TMAX= 563.02 ANGLE= 26.7
BOTT: SMAX= 3143.47 SMIN= 2473.83 TMAX= 334.82 ANGLE= -82.2

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

MAXIMUM PRINCIPAL SHEAR VONMISES TRESCA
PRINCIPAL STRESS STRESS STRESS STRESS
3.664812E+03 -3.704341E+03 5.630193E+02 3.305413E+03 3.704341E+03

PLATE NO. 47 47 47 47 47
CASE NO. 1 1 3 1 1

**************************************************************************
84. START CONCRETE DESIGN
STAAD SPACE -- PAGE NO. 7
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
CONCRETE DESIGN

85. CODE BS8007
PROGRAM CODE REVISION V1.0_8007_87/1
86. DESIGN ELEMENT 47 --
STAAD SPACE -- PAGE NO. 8
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
**Application Examples**

**EX. British Design Examples**

---

**ELEMENT DESIGN TO BS8007 AND BS8110**

**ELEMENT NO. 47**

![Diagram](image)

- **A >**
- **k**
- **o**
- **g**
- **My**
- **Section A-A**

**Depth=150 mm Width=1000 mm Cover=20 mm**

---

**Ultimate Limit State**

<table>
<thead>
<tr>
<th>Max. Moment kNm/m</th>
<th>12 mm Bars</th>
<th>16 mm Bars</th>
<th>20 mm Bars</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mx Top</td>
<td>0.0</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>Mx Bot</td>
<td>-10.4</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>My Top</td>
<td>0.0</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>My Bot</td>
<td>-13.3</td>
<td>200</td>
<td>200</td>
</tr>
</tbody>
</table>

---

**SERVICEABILITY LIMIT STATE**

<table>
<thead>
<tr>
<th>Longitudinal Moment Mx kNm/m</th>
<th>Transverse Moment My kNm/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top 0.0 L. 0 Bot -10.4 L. 1</td>
<td>Top 0.0 L. 0 Bot -13.3 L. 1</td>
</tr>
</tbody>
</table>

---

**Thermal Crack Width Calculations Based On Wmax=Smax*R*T1*Alfa**

**Surface Type:** Suspended  
**Construction type:** 1  
**Temp. Range:** 30°C

---

**END OF ELEMENT DESIGN**

---

* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/*

---

**STAAD SPACE**

---

**EXAMPLE PROBLEM WITH FRAME MEMBERS AND**

---

**END OF THE STAAD.Pro RUN**

---

**DATE= APR 14,2019 TIME= 22:55: 1****

---

* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/*

---

**STAAD.Pro**

4715  
**User Manual**
EX. UK-10 Finite Element Model for a Rectangular Tank

A tank structure is modeled with four-noded plate elements. Water pressure from inside is used as loading for the tank. Reinforcement calculations have been done for some elements.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-10 Finite Element Model for a Rectangular Tank.STD when you install the program.
Figure 493: Deflected Shape

Actual input is shown in bold lettering followed by explanation.

STAAD SPACE FINITE ELEMENT MODEL OF TANK
* STRUCTURE

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D) structure.

UNITS METER KNS

Defines the input units for the data that follows.

JOINT COORDINATES
1 0. 0. 0. 5 0. 6. 0.
REPEAT 4 1.5. 0. 0.
REPEAT 4 0. 0. 1.5.
REPEAT 4 -1.5. 0. 0.
REPEAT 3 0. 0. -1.5.
81 1.5. 0. 1.5. 83 1.5. 0. 4.5.
REPEAT 2 1.5. 0. 0.

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command generates joint coordinates by repeating the pattern of the previous line of joint coordinates. The number following the REPEAT command is the number of repetitions to be carried out. This is followed by X, Y and Z coordinate increments. See section 5.11 of the Technical Reference Manual (on page 2425).

ELEMENT INCIDENCES
1 1 2 7 6 TO 4 1 1
REPEAT 14 4 5
61 76 77 2 1 TO 64 1 1
65 1 6 81 76
66 76 81 82 71
67 71 82 83 66
68 66 83 56 61
69 6 11 84 81
70 81 84 85 82
71 82 85 86 83
72 83 86 51 56
73 11 16 87 84
74 84 87 88 85
75 85 88 89 86
76 86 89 46 51
77 16 21 26 87
Element connectivities are input as above by providing the element number followed by joint numbers defining the element. The REPEAT command generates element incidences by repeating the pattern of the previous line of element nodes. The number following the REPEAT command is the number of repetitions to be carried out and that is followed by element and joint number increments. See section 5.12 of the Technical Reference Manual (on page 2428).

```
UNIT MMS
ELEMENT PROPERTIES
1 TO 80 TH 200.0
```

Element properties are provided by specifying that the elements are 200.0 mm THick.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
  E 21.0
  POISSON 0.17
  DENSITY 2.4e-008
  ALPHA 5e-006
  DAMP 0.05
  G 9.25
  TYPE CONCRETE
  STRENGTH FCU 0.028
END DEFINE MATERIAL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
SUPPORT
  1 TO 76 BY 5 81 TO 89 PINNED
```

Pinned supports are specified at the joints listed above. No moments will be carried by these supports. The expression 1 TO 76 BY 5 means 1, 6, 11, etc. up to 76.

```
UNIT METER
LOAD 1
  ELEMENT LOAD
  4 TO 64 BY 4 PR 50.0
  3 TO 63 BY 4 PR 100.0
  2 TO 62 BY 4 PR 150.0
  1 TO 61 BY 4 PR 200.0
```

Load case 1 is initiated. It consists of element loads in the form of uniform pressure (indicated by PR) acting along the local z-axis.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
UNIT MMS
  PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
  PRINT ELEM FORCE LIST 9 TO 16
```
Joint displacements for a selected set of nodes and element corner forces for some elements are written in the output file as a result of the above commands. The forces printed are in the global directions at the nodes of the elements. The LIST option restricts the print output to that for the joints/elements listed.

```
START CONCRETE DESIGN
```

The above command initiates concrete design.

```
CODE BRITISH
DESIGN SLAB 9 12
```

Slabs (i.e., elements) 9 and 12 will be designed and the reinforcement requirements obtained. In STAAD, elements are typically designed for the moments MX and MY at the centroid of the element.

```
END CONCRETE DESIGN
```

Terminates the concrete design operation.

```
FINISH
```

This command terminates the STAAD run.

### Input File

```
STAAD SPACE FINITE ELEMENT MODEL OF TANK STRUCTURE
UNIT METER KNS
JOINT COORDINATES
  1 0.0 0.0 0.0 5 0.0 6.0 0.0
REPEAT 4 1.5 0.0 0.0
REPEAT 4 0.0 0.0 1.5
REPEAT 4 -1.5 0.0 0.0
REPEAT 3 0.0 0.0 -1.5
  81 1.5 0.0 1.5 83 1.5 0.0 4.5
REPEAT 2 1.5 0.0 0.0
ELEMENT INCIDENCES
  1 1 2 7 6 TO 4 1 1
REPEAT 14 4 5
  61 76 77 2 1 TO 64 1 1
  65 1 6 81 76
  66 76 81 82 71
  67 71 82 83 66
  68 66 83 56 61
  69 6 11 84 81
  70 81 84 85 82
  71 82 85 86 83
  72 83 86 51 56
  73 11 16 87 84
  74 84 87 88 85
  75 85 88 89 86
  76 86 89 46 51
  77 16 21 26 87
  78 87 26 31 88
  79 88 31 36 89
  80 89 36 41 46
UNIT MMS
ELEMENT PROPERTIES
  1 TO 80 TH 200.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
```
POISSON 0.17
DENSITY 2.4e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.028
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 76 BY 5 81 TO 89 PINNED
UNIT METER
LOAD 1
ELEMENT LOAD
4 TO 64 BY 4 PR 50.0
3 TO 63 BY 4 PR 100.0
2 TO 62 BY 4 PR 150.0
1 TO 61 BY 4 PR 200.0
PERFORM ANALYSIS
UNIT MMS
PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
PRINT ELEM FORCE LIST 9 TO 16
START CONCRETE DESIGN
CODE BS8007
DESIGN SLAB 9 12
END CONCRETE DESIGN
FINISH

STAAD Output File

1. STAAD SPACE FINITE ELEMENT MODEL OF TANK STRUCTURE
INPUT FILE: UK-10 Finite Element Model for a Rectangular Tank.STD
2. UNIT METER KNS
3. JOINT COORDINATES
   1 0.0 0.0 0.0 5 0.0 6.0 0.0
   2 1.5 0.0 0.0
   3 0.0 0.0 1.5
   4 REPEAT 4 -1.5 0.0 0.0
   5 REPEAT 3 0.0 0.0 -1.5
   6 81 1.5 0.0 1.5 83 1.5 0.0 4.5
   7 REPEAT 2 1.5 0.0 0.0
11. ELEMENT INCIDENCES
   12. 1 1 2 7 6 TO 4 1 1
   13. REPEAT 14 4 5
   14. 61 76 77 2 1 TO 64 1 1

****************************************************
*                                                  *
*           STAAD.Pro CONNECT Edition              *
*           Version  22.01.00.**                   *
*           Proprietary Program of                 *
*           Bentley Systems, Inc.                  *
*           Date=    APR 14, 2019                  *
*           Time=    22:52:42                      *
*                                                  *
*  Licensed to: Bentley Systems Inc                *
****************************************************
15. 65 1 6 81 76
16. 66 76 81 82 71
17. 67 71 82 83 66
18. 68 66 83 56 61
19. 69 6 11 84 81
20. 70 81 84 85 82
21. 71 82 85 86 83
22. 72 83 86 51 56
23. 73 11 16 87 84
24. 74 84 87 88 85
25. 75 85 88 89 86
26. 76 86 89 46 51
27. 77 16 21 26 87
28. 78 87 26 31 88
29. 79 88 31 36 89
30. 80 89 36 41 46
31. UNIT MMS
32. ELEMENT PROPERTIES
33. 1 TO 80 TH 200.0
34. DEFINE MATERIAL START
35. ISOTROPIC CONCRETE
36. E 21.0
37. POISSON 0.17
38. DENSITY 2.4E-008
39. ALPHA 5E-006
40. DAMP 0.05
41. G 9.25
42. TYPE CONCRETE
43. STRENGTH FCU 0.028
44. END DEFINE MATERIAL
45. CONSTANTS
46. MATERIAL CONCRETE ALL
47. SUPPORT
48. 1 TO 76 BY 5 81 TO 89 PINNED
49. UNIT METER
50. LOAD 1
51. ELEMENT LOAD
52. 4 TO 64 BY 4 PR 50.0
53. 3 TO 63 BY 4 PR 100.0
54. 2 TO 62 BY 4 PR 150.0
55. 1 TO 61 BY 4 PR 200.0
56. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS 89
NUMBER OF MEMBERS 0
NUMBER OF PLATES 80
NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0
NUMBER OF SUPPORTS 25

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 1, TOTAL DEGREES OF FREEDOM = 459
TOTAL LOAD COMBINATION CASES = 0 SO FAR.
57. UNIT MMS
58. PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
JOINT DISPLACE LIST 5
FINITE ELEMENT MODEL OF TANK STRUCTURE -- PAGE NO. 3
JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE
-----------------------------------
## Application Examples

### EX. British Design Examples

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>-0.0103</td>
<td>0.0007</td>
<td>-0.0103</td>
<td>0.0002</td>
<td>-0.0002</td>
</tr>
<tr>
<td>25</td>
<td>1</td>
<td>0.0103</td>
<td>0.0007</td>
<td>-0.0103</td>
<td>0.0002</td>
<td>0.0002</td>
</tr>
<tr>
<td>45</td>
<td>1</td>
<td>0.0103</td>
<td>0.0007</td>
<td>0.0103</td>
<td>-0.0002</td>
<td>0.0002</td>
</tr>
<tr>
<td>65</td>
<td>1</td>
<td>-0.0103</td>
<td>0.0007</td>
<td>0.0103</td>
<td>-0.0002</td>
<td>0.0002</td>
</tr>
</tbody>
</table>

*************** END OF LATEST ANALYSIS RESULT ***************

59. PRINT ELEM FORCE LIST 9 TO 16

**FINITE ELEMENT MODEL OF TANK STRUCTURE**

---

### Global Corner Forces

<table>
<thead>
<tr>
<th>JOINT</th>
<th>FX</th>
<th>FY</th>
<th>FZ</th>
<th>MX</th>
<th>MY</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELE.NO. 9 FOR LOAD CASE 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-8.8840E+01</td>
<td>-5.4966E+01</td>
<td>1.8236E+02</td>
<td>1.3176E+05</td>
<td>-4.1177E+04</td>
<td>1.0696E+04</td>
</tr>
<tr>
<td>12</td>
<td>-8.2763E+01</td>
<td>-6.0241E+00</td>
<td>-1.5662E+02</td>
<td>7.9554E+04</td>
<td>5.9496E+04</td>
<td>-1.6817E+04</td>
</tr>
<tr>
<td>17</td>
<td>1.4375E+02</td>
<td>4.7401E+01</td>
<td>-1.6873E+02</td>
<td>1.2776E+05</td>
<td>-1.1029E+05</td>
<td>2.1861E+04</td>
</tr>
<tr>
<td>16</td>
<td>2.7850E+01</td>
<td>1.3590E+01</td>
<td>1.4300E+02</td>
<td>1.4896E+05</td>
<td>5.3365E+04</td>
<td>-1.5740E+04</td>
</tr>
</tbody>
</table>

| ELE.NO. 10 FOR LOAD CASE 1 |
| 12 | -1.8729E+02 | 6.0241E+00 | -4.0257E+01 | -7.9554E+04 | 1.0833E+05 | 4.7435E+04 |
| 13 | -1.9921E+02 | -3.6766E+01 | -6.4125E+01 | 6.9592E+04 | 1.0234E+05 | -5.8037E+04 |
| 18 | 2.2995E+02 | -1.3963E+01 | 3.0505E+01 | 5.8552E+04 | -5.3896E+04 | 6.1870E+04 |
| 17 | 1.5655E+02 | 4.4705E+01 | 7.3877E+01 | 1.8395E+03 | -2.0026E+02 | -5.1268E+04 |

| ELE.NO. 11 FOR LOAD CASE 1 |
| 13 | -2.1146E+02 | 3.6766E+01 | -7.6500E+01 | -6.9592E+04 | 1.2194E+05 | 5.7366E+04 |
| 14 | -2.3180E+02 | -1.4769E+01 | -2.4156E+01 | 2.6339E+04 | 1.0470E+05 | -6.4754E+04 |
| 19 | 2.0981E+02 | -3.2909E+01 | 8.3838E+01 | -1.5592E+04 | -2.4195E+04 | 6.5369E+04 |
| 18 | 2.3345E+02 | 1.0912E+01 | 1.6817E+01 | -3.0678E+04 | -5.1461E+04 | -5.7982E+04 |

| ELE.NO. 12 FOR LOAD CASE 1 |
| 14 | -1.7545E+02 | 1.4769E+01 | -6.0219E+01 | -2.6339E+04 | 1.0553E+04 | 4.3931E+04 |
| 15 | -1.7917E+02 | -1.7164E-07 | -2.8125E+01 | 5.4335E-04 | 9.5913E+04 | -4.9179E+04 |
| 20 | 1.6440E+02 | -2.5971E+00 | 6.5505E+01 | -2.8746E+04 | -2.4521E+04 | 6.5369E+04 |
| 19 | 1.9022E+02 | -1.2172E+01 | 2.2839E+01 | -9.8543E+02 | -4.4410E+04 | -5.7982E+04 |

| ELE.NO. 13 FOR LOAD CASE 1 |
| 15 | -1.0576E+02 | -8.2837E+01 | -1.7234E+01 | -1.4733E+04 | 2.7047E+04 | 1.5133E+04 |
| 17 | 3.2926E+01 | 6.8334E+01 | 7.6979E+01 | -8.9147E+03 | -4.6371E+03 | -2.4466E+04 |
| 22 | 1.8410E+02 | 2.6957E+01 | 8.1630E+01 | -1.6851E+04 | 6.2837E+04 | 2.9884E+04 |
| 21 | -4.5408E+01 | 1.2421E+02 | 3.1252E+02 | 3.5322E+04 | 5.6274E+04 | -2.0551E+04 |

| ELE.NO. 14 FOR LOAD CASE 1 |
| 17 | -2.6737E+02 | -2.3772E+01 | -2.2191E+02 | -1.2068E+05 | 1.1453E+05 | 5.3873E+04 |
| 18 | -2.5271E+02 | -5.3156E+00 | -1.4927E+02 | 3.2784E+04 | 5.0765E+04 | -5.2692E+04 |
| 23 | 2.8179E+02 | 5.6045E+01 | 2.2731E+02 | -8.0772E+04 | 2.0374E+05 | 6.3464E+04 |
| 22 | 2.3828E+02 | -2.6957E+01 | 1.4388E+02 | 5.1613E+04 | 1.8801E+05 | -4.6645E+04 |

| ELE.NO. 15 FOR LOAD CASE 1 |
| 19 | -2.1070E+02 | 8.3668E+00 | -1.7930E+02 | -6.0658E+04 | 5.4592E+04 | 4.8803E+04 |
| 18 | -2.3137E+02 | 2.9518E+01 | -1.8160E+01 | -1.6505E+02 | 7.1971E+04 | 4.9794E+04 |
| 24 | 2.4858E+02 | 5.6045E+01 | 1.6244E+02 | 6.3464E+04 | 1.9404E+05 | -4.6155E+04 |

| ELE.NO. 16 FOR LOAD CASE 1 |
| 19 | -1.6865E+02 | 1.5563E+01 | -1.2724E+02 | -2.7116E+04 | 2.1920E+04 | 2.7200E+04 |
| 20 | -2.6404E+02 | 2.5971E+00 | -1.2176E+02 | 2.8746E+04 | 4.2521E+04 | -4.5374E+04 |
| 25 | 1.4624E+02 | -1.1240E+01 | 1.8121E+01 | -5.5416E+04 | 1.6204E+05 | 5.5416E+04 |
| 24 | 1.8681E+02 | -1.8160E+01 | 1.3088E+02 | 5.9238E+04 | 1.6501E+05 | -3.7241E+04 |

60. START CONCRETE DESIGN

**FINITE ELEMENT MODEL OF TANK STRUCTURE**

---

61. CODE BS8007

PROGRAM CODE REVISION V1.0_8007_87/1
### DESIGN SLAB 9 12

**FINITE ELEMENT MODEL OF TANK STRUCTURE**

#### ELEMENT DESIGN TO BS8007 AND BS8110

<table>
<thead>
<tr>
<th>ELEMENT NO. 9</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>A &gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>l--------k</td>
</tr>
<tr>
<td>! y</td>
</tr>
<tr>
<td>z+ --&gt; x</td>
</tr>
<tr>
<td>1--------j</td>
</tr>
</tbody>
</table>

**Depth=200 mm  Width=1000 mm  Cover=20 mm**

**Ultimate Limit State**

<table>
<thead>
<tr>
<th>Ultimate Limit State</th>
<th>12 mm Bars</th>
<th>16 mm Bars</th>
<th>20 mm Bars</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max. Moment (kNm/m)</td>
<td>C/C AS R. AS P.</td>
<td>C/C AS R. AS P.</td>
<td>C/C AS R. AS P.</td>
</tr>
</tbody>
</table>

| Mx Top    | 0.0 | 0   | 200 | 259 | 565 | 200 | 259 | 1065 | 200 | 259 | 1572 |
| Mx Bot    | -24.5 | 1   | 200 | 367 | 565 | 200 | 372 | 1065 | 200 | 376 | 1572 |
| My Top    | 25.1 | 1   | 200 | 404 | 565 | 200 | 420 | 1065 | 200 | 437 | 1572 |
| My Bot    | 0.0 | 0   | 200 | 259 | 565 | 200 | 259 | 1065 | 200 | 259 | 1572 |

**SERVICEABILITY LIMIT STATE**

<table>
<thead>
<tr>
<th>Longitudinal Moments Mx (kNm/m)</th>
<th>Transverse Moments My (kNm/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top= 0.0 L. 0</td>
<td>Bot= -24.5 L. 1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>12</th>
<th>16</th>
<th>20</th>
<th>@</th>
<th>12</th>
<th>16</th>
<th>20</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>0.00</td>
<td>0.00 e</td>
<td>0.08</td>
<td>0.04</td>
<td>0.03 0.17</td>
<td>0.12</td>
</tr>
<tr>
<td>0.00</td>
<td>0.00</td>
<td>0.00 f</td>
<td>0.20</td>
<td>0.11</td>
<td>0.08 0.27</td>
<td>0.17</td>
</tr>
<tr>
<td>0.00</td>
<td>0.00</td>
<td>0.00 g</td>
<td>0.10</td>
<td>0.06</td>
<td>0.05 0.13</td>
<td>0.08</td>
</tr>
</tbody>
</table>

**Thermal Crack Width Calculations Based On Wmax=Smax*R*T1*Alfa**

<table>
<thead>
<tr>
<th>Surface Type</th>
<th>Suspended</th>
<th>Construction type</th>
<th>1</th>
<th>Temp. Range = 30 C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface Zones &amp; ROWcrit</td>
<td>8 mm bars</td>
<td>10 mm Bars</td>
<td>12 mm Bars</td>
<td>16 mm Bars</td>
</tr>
<tr>
<td>Top : 100 mm 350 mm2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Smax</td>
<td>mm</td>
<td>765</td>
<td>765</td>
<td>957</td>
</tr>
<tr>
<td>Wmax</td>
<td>mm</td>
<td>0.14</td>
<td>0.14</td>
<td>0.17</td>
</tr>
<tr>
<td>Sp. For Wmax = 0.20 mm</td>
<td>208</td>
<td>208</td>
<td>260</td>
<td>260</td>
</tr>
</tbody>
</table>

### APPLICATION EXAMPLES

**EX. British Design Examples**

STAAD.Pro User Manual
**Application Examples**

**EX. British Design Examples**

---

**Ultimate Limit State**

<table>
<thead>
<tr>
<th>Ultimate Limit State</th>
<th>12 mm Bars</th>
<th>16 mm Bars</th>
<th>20 mm Bars</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mx Top = 0.0</td>
<td>200</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>Mx Bot = -0.5</td>
<td>200</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>My Top = 90.1</td>
<td>1621</td>
<td>100</td>
<td>150</td>
</tr>
<tr>
<td>My Bot = 0.0</td>
<td>200</td>
<td>200</td>
<td>200</td>
</tr>
</tbody>
</table>

---

**Serviceability Limit State**

<table>
<thead>
<tr>
<th>Serviceability Limit State</th>
<th>Element No. 12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Moments Mx</td>
<td>Transverse Moments My</td>
</tr>
<tr>
<td>Flexural Crack Width mm</td>
<td>Flexural Crack Width mm</td>
</tr>
<tr>
<td>Top = 0.0 L. 0</td>
<td>Bot = -0.5 L. 1</td>
</tr>
<tr>
<td>12</td>
<td>16</td>
</tr>
<tr>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>e</td>
<td>-0.03</td>
</tr>
<tr>
<td>f</td>
<td>-0.08</td>
</tr>
<tr>
<td>g</td>
<td>-0.04</td>
</tr>
</tbody>
</table>

---

**Surface Type:** Suspended  
**Construction Type:** 1  
**Temp. Range:** 30°C

**Surface Zones & ROWcrit**

<table>
<thead>
<tr>
<th>Surface Zones &amp; ROWcrit</th>
<th>8 mm bars</th>
<th>10 mm Bars</th>
<th>12 mm Bars</th>
<th>16 mm Bars</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top: 100 mm 350 mm²</td>
<td>Top: 100 mm 350 mm²</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bot: 100 mm 350 mm²</td>
<td>Bot: 100 mm 350 mm²</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**Thermal Crack Width Calculations Based On Wmax=Smax*R*T1*Alfa**

---

**END OF ELEMENT DESIGN**
EX. UK-11 Response Spectrum Analysis of a Frame

Dynamic analysis (Response Spectrum) is performed for a steel structure. Results of a static and dynamic analysis are combined. The combined results are then used for steel design.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-11 Response Spectrum Analysis of a Frame.STD when you install the program.

Where:

- \( L_1 = 3 \, \text{m} \), \( L_2 = 6 \, \text{m} \)
- \( w = 22.5 \, \text{kN/m} \)
- \( P_1 = 25 \, \text{kN} \), \( P_2 = 37.5 \, \text{kN} \)

Actual input is shown in bold lettering followed by explanation.

**STAAD PLANE RESPONSE SPECTRUM ANALYSIS**

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

**UNIT METER KNS**
Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 6.0 0.0 0.0 
3 0.0 3.0 0.0 ; 4 6.0 3.0 0.0
5 0.0 6.0 0.0 ; 6 6.0 6.0 0.0
```

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are all the same, in this case, zeros.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6 
5 3 4 ; 6 5 6
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTIES BRITISH
1 TO 4 TA ST UC254X254X73 
5 TA ST UB305X165X54 
6 TA ST UB203X133X30
```

Properties for all members are assigned from the British steel table. The word ST stands for standard single section.

```
SUPPORTS
1 2 FIXED
```

Fixed supports are specified at joints 1 and 2.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

Material constants such as E (modulus of elasticity), Poisson’s ratio and density (DEN) are specified above. Length unit is changed from METER to MMS to facilitate the input.

```
CUT OFF MODE SHAPE 2
```

The number of mode shapes to be considered in dynamic analysis is set to 2. Without the above command, this will be set to the default. [See section 5.30.1 of the Technical Reference Manual](on page 2539).

```
* LOAD 1 WILL BE STATIC LOAD
UNIT METER
LOAD 1 DEAD AND LIVE LOADS
SELFWEIGHT Y -1.0
```

Load case 1 is initiated followed by a title. Prior to this, the length unit is changed to METER for specifying distributed member loads. A line starting with an asterisk (*) mark indicates a comment line.
The above command indicates that the selfweight of the structure acting in the global Y direction is part of this load case. The factor of -1.0 is meant to indicate that the load acts opposite to the positive direction of global Y, hence downwards.

Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The word UNI stands for uniformly distributed load while CON stands for concentrated load. GY is followed by the value of the load and the distance at which it is applied.

The two lines which begin with the asterisk are comment lines which tell us the purpose of the next load case. Load case 2 is then initiated along with an optional title. This will be a dynamic load case. Permanent masses will be provided in the form of loads. These masses (in terms of loads) will be considered for the eigensolution. Internally, the program converts these loads to masses, hence it is best to specify them as absolute values (without a negative sign). Also, the direction (X, Y, Z etc.) of the loads will correspond to the dynamic degrees of freedom in which the masses are capable of vibrating. In a PLANE frame, only X and Y directions need to be considered. In a SPACE frame, masses (loads) should be provided in all three (X, Y and Z) directions if they are active along all three. The user has the freedom to restrict one or more directions.

The above commands indicate that the selfweight of the structure acting in the global X and Y directions with a factor of 1.0 is taken into consideration for the mass matrix.

The mass matrix will also consist of terms derived from the above member loads. GX and GY indicate that the load, and hence the resulting mass, is capable of vibration along the global X and Y directions. The word CON stands for concentrated load. Concentrated forces of 25, 37.5, and 25 kNs are located at 1.8m, 3.0m and 4.2m from the start of member 5.

The SPECTRUM command specifies a Eurocode 8 2004 seismic response spectrum load. The modal responses will be combined using the CQC method. Here, the spectrum effect is in the global X direction with a factor of 1.0. EC8 2004 response spectra are always given in terms of acceleration (ACC). A damping ratio of 0.05 (5%) is used. The second line then gives the soil type along with the alpha ratio and behavior factor, Q.
In a response spectrum analysis, the sign of the forces cannot be determined, and hence are absolute numbers. Consequently, to account for the fact that the force could be positive or negative, it is necessary to create 2 load combination cases. That is what is being done above. Load combination case no. 3 consists of the sum of the static load case (1) with the positive direction of the dynamic load case (2). Load combination case no. 4 consists of the sum of the static load case (1) with the negative direction of the dynamic load case (2). In both cases, the result is factored by 0.75.

**PERFORM ANALYSIS PRINT MODE SHAPES**

This command instructs the program to proceed with the analysis. The PRINT command instructs the program to print mode shape values.

**PRINT ANALYSIS RESULTS**

Displacements, reactions and member forces are recorded in the output file using the above command.

**LOAD LIST 1 3 4**
**PARAMETER**
**CODE EN 1993-1-1:2005**
**NA 1**
**SELECT ALL**

A steel design in the form of a member selection is performed based on the rules of the Eurocode 8 code using the UK national annex. Only the member forces resulting from load cases 1, 3 and 4 will be considered for these calculations.

**FINISH**

This command terminates the STAAD run.

**Input File**

```
STAAD PLANE RESPONSE SPECTRUM ANALYSIS
UNIT METER KNS
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 6.0 0.0 0.0
3 0.0 3.0 0.0 ; 4 6.0 3.0 0.0
5 0.0 6.0 0.0 ; 6 6.0 6.0 0.0
MEMBER INCIDENCES
1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
5 3 4 ; 6 5 6
MEMBER PROPERTIES BRITISH
1 TO 4 TA ST UC254X254X73
5 TA ST UB305X165X54
6 TA ST UB203X133X30
SUPPORTS
1 2 FIXED
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```
CUT OFF MODE SHAPE 2
*LOAD 1 WILL BE STATIC LOAD
UNIT METER
LOAD 1 DEAD AND LIVE LOADS
SELFWEIGHT Y -1.0
MEMBER LOADS
5 CON GY -25.0 1.8
5 CON GY -37.5 3.0
5 CON GY -25.0 4.2
5 6 UNI Y -22.5
* NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
* WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 2 SEISMIC LOADING
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
MEMBER LOADS
5 CON GX 25.0 1.8
5 CON GX 25.0 1.8
5 CON GX 37.5 3.0
5 CON GX 25.0 4.2
5 6 UNI Y 22.5
SPECTRUM CQC EURO X 1.0 ACC DAMP 0.05
SOIL TYPE C ALPHA 0.15 Q 1.5
LOAD COMBINATION 3
1 0.75 2 0.75
LOAD COMBINATION 4
1 0.75 2 -0.75
PERFORM ANALYSIS PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
LOAD LIST 1 3 4
PARAMETER
CODE EN 1993-1-1:2005
NA 1
SELECT ALL
FINISH

STAAD Output File

************************************************************************************
* STAAD.Pro CONNECT Edition
* Version 22.01.00.**
* Proprietary Program of
* Bentley Systems, Inc.
* Date= APR 14, 2019
* Time= 22:52:47
* Licensed to: Bentley Systems Inc
************************************************************************************

1. STAAD PLANE RESPONSE SPECTRUM ANALYSIS
INPUT FILE: UK-11 Response Spectrum Analysis of a Frame.STD
2. UNIT METER KNS
3. JOINT COORDINATES
4. 1 0.0 0.0 0.0 ; 2 6.0 0.0 0.0
5. 3 0.0 3.0 0.0 ; 4 6.0 3.0 0.0
6. 5 0.0 6.0 0.0 ; 6 6.0 6.0 0.0
7. MEMBER INCIDENCES
8. 1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
9. 5 3 4 ; 6 5 6
10. MEMBER PROPERTIES BRITISH
11. 1 TO 4 TA ST UC254X254X73
12. 5 TA ST UB305X165X54
13. 6 TA ST UB203X133X30
14. SUPPORTS
15. 1 2 FIXED
16. UNIT MMS
17. DEFINE MATERIAL START
18. ISOTROPIC STEEL
19. E 210
20. POISSON 0.3
21. DENSITY 7.6977E-008
22. ALPHA 6E-006
23. DAMP 0.03
24. TYPE STEEL
25. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
26. END DEFINE MATERIAL
27. CONSTANTS
28. MATERIAL STEEL ALL
29. CUT OFF MODE SHAPE 2
30. *LOAD 1 WILL BE STATIC LOAD
31. UNIT METER
32. LOAD 1 DEAD AND LIVE LOADS
33. SELFWEIGHT Y -1.0
34. MEMBER LOADS
35. 5 CON GY -25.0 1.8
36. 5 CON GY -37.5 3.0
37. 5 CON GY -25.0 4.2
38. 5 6 UNI Y -22.5
39. RESPONSE SPECTRUM ANALYSIS
-- PAGE NO. 2
40. * NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
41. * WITH MASSES PROVIDED IN TERMS OF LOAD.
42. LOAD 2 SEISMIC LOADING
43. SELFWEIGHT X 1.0
44. SELFWEIGHT Y 1.0
45. MEMBER LOADS
46. 5 CON GX 25.0 1.8
47. 5 CON GX 25.0 1.8
48. 5 CON GX 37.5 3.0
49. 5 CON GX 37.5 3.0
50. 5 CON GX 25.0 4.2
51. SPECTRUM CQC EURO X 1.0 ACC DAMP 0.05
52. SOIL TYPE C ALPHA 0.15 Q 1.5
53. LOAD COMBINATION 3
54. 1 0.75 2 0.75
55. LOAD COMBINATION 4
56. 1 0.75 2 -0.75
57. PERFORM ANALYSIS PRINT MODE SHAPES

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS       6
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 12
TOTAL LOAD COMBINATION CASES = 2 SO FAR.
***NOTE: MASSES DEFINED UNDER LOAD# 2 WILL FORM THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
EIGEN METHOD : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED = 2
NUMBER OF EXISTING MASSES IN THE MODEL = 8
NUMBER OF MODES THAT WILL BE USED = 2
*** EIGENSOLUTION : ADVANCED METHOD ***
RESPONSE SPECTRUM ANALYSIS -- PAGE NO. 3
CALCULATED FREQUENCIES FOR LOAD CASE 2
MODE FREQUENCY(CYCLES/SEC) PERIOD(SEC)
1 5.178 0.19312
2 19.435 0.05145
RESPONSE SPECTRUM ANALYSIS -- PAGE NO. 4
MODE SHAPES
-----------
JOINT MODE X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN
1 1 0.00000 0.00000 0.00000 0.000E+00 0.000E+00 0.000E+00
2 1 0.00000 0.00000 0.00000 0.000E+00 0.000E+00 0.000E+00
3 1 0.58345 0.00218 0.00000 0.000E+00 0.000E+00 -3.862E-03
4 1 0.58345 -0.00218 0.00000 0.000E+00 0.000E+00 -3.862E-03
5 1 1.00000 0.00251 0.00000 0.000E+00 0.000E+00 -2.794E-03
6 1 1.00000 -0.00251 0.00000 0.000E+00 0.000E+00 -2.794E-03
MODE SHAPES
-----------
JOINT MODE X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN
1 2 0.00000 0.00000 0.00000 0.000E+00 0.000E+00 0.000E+00
2 2 0.00000 0.00000 0.00000 0.000E+00 0.000E+00 0.000E+00
3 2 -0.07065 0.00282 0.00000 0.000E+00 0.000E+00 -2.798E-03
4 2 -0.07065 -0.00282 0.00000 0.000E+00 0.000E+00 -2.798E-03
5 2 1.00000 0.00400 0.00000 0.000E+00 0.000E+00 -9.739E-03
6 2 1.00000 -0.00400 0.00000 0.000E+00 0.000E+00 -9.739E-03
RESPONSE SPECTRUM LOAD 2
RESPONSE LOAD CASE 2
MODAL WEIGHT (MODAL MASS TIMES g) IN KNS GENERALIZED
MODE X Y Z WEIGHT
1 9.711083E+01 9.577115E-18 0.000000E+00 3.624621E+01
2 1.780698E+00 1.662500E-16 0.000000E+00 4.389139E+00
CQC MODAL COMBINATION METHOD USED.
DYNAMIC WEIGHT X Y Z 9.889183E+01 9.889183E+01 0.000000E+00 KNS
MISSING WEIGHT X Y Z -3.075424E-04 -9.889183E+01 0.000000E+00 KNS
MODAL WEIGHT X Y Z 9.889153E+01 1.758271E-16 0.000000E+00 KNS
RESPONSE SPECTRUM ANALYSIS -- PAGE NO. 5
MODE ACCELERATION-G DAMPING
1 0.22191 0.05000
2 0.15815 0.05000
MODAL BASE ACTIONS FORCES IN KNS LENGTH IN METER
MODE PERIOD FX FY FZ MX MY
MZ 1 0.193 21.55 0.00 0.00 0.00 0.00

Application Examples
EX. British Design Examples
STAAD.Pro 4731 User Manual
### PARTICIPATION FACTORS

**MASS PARTICIPATION FACTORS IN PERCENT**

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>98.20</td>
<td>0.00</td>
<td>0.00</td>
<td>98.199</td>
<td>0.000</td>
<td>0.000</td>
<td>21.55</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1.80</td>
<td>0.00</td>
<td>0.00</td>
<td>100.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.28</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**BASE SHEAR IN KNS**

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>21.55</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**TOTAL SRSS SHEAR**: 21.55 KNS

**TOTAL 10PCT SHEAR**: 21.55 KNS

**TOTAL ABS SHEAR**: 21.83 KNS

**TOTAL CQC SHEAR**: 21.55 KNS

---

***WARNING: NO RIGID FLOOR DIAPHRAGM EXISTS FOR THE STRUCTURE.***

**RESULTS ON EARTHQUAKE MODE IN USER-INTERFACE MAY BE APPROXIMATE.**

**WARNING : NO WELL DEFINED FLOOR LEVEL EXISTS FOR "STAAD SPACE" MODEL.**

**CALCULATION OF STOREY SHEAR DUE TO MISSING MASS OR TORSION IGNORED.**

---

### RESPONSE SPECTRUM ANALYSIS

**JOINT DISPLACEMENT (CM RADIANS)**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>4</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>4</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-0.0642</td>
<td>-0.0283</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0017</td>
</tr>
<tr>
<td>2</td>
<td>0.1963</td>
<td>0.0007</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0005</td>
</tr>
<tr>
<td>3</td>
<td>0.1441</td>
<td>-0.0287</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0009</td>
</tr>
<tr>
<td>4</td>
<td>-0.1564</td>
<td>-0.0218</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0017</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.0642</td>
<td>-0.0283</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0017</td>
</tr>
<tr>
<td>2</td>
<td>0.1963</td>
<td>0.0007</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0005</td>
</tr>
<tr>
<td>3</td>
<td>0.1564</td>
<td>-0.0287</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0017</td>
</tr>
<tr>
<td>4</td>
<td>-0.1441</td>
<td>-0.0218</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0009</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.0163</td>
<td>0.0390</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0017</td>
</tr>
<tr>
<td>2</td>
<td>0.3366</td>
<td>0.0088</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0004</td>
<td>0.0004</td>
</tr>
<tr>
<td>3</td>
<td>0.2647</td>
<td>-0.0286</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0010</td>
</tr>
<tr>
<td>4</td>
<td>-0.2402</td>
<td>-0.0299</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0015</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-0.0163</td>
<td>0.0390</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0017</td>
</tr>
<tr>
<td>2</td>
<td>0.3366</td>
<td>0.0088</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0004</td>
<td>0.0004</td>
</tr>
<tr>
<td>3</td>
<td>0.2402</td>
<td>-0.0286</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0015</td>
</tr>
<tr>
<td>4</td>
<td>-0.2647</td>
<td>-0.0299</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0010</td>
</tr>
</tbody>
</table>

---

**SUPPORT REACTIONS -UNIT KNS METE**

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>23.47</td>
<td>185.52</td>
<td>0.00</td>
<td>0.00</td>
<td>-21.57</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>10.78</td>
<td>4.78</td>
<td>0.00</td>
<td>0.00</td>
<td>20.25</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>25.69</td>
<td>142.72</td>
<td>0.00</td>
<td>0.00</td>
<td>-1.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>9.52</td>
<td>135.56</td>
<td>0.00</td>
<td>0.00</td>
<td>-31.37</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-23.47</td>
<td>185.52</td>
<td>0.00</td>
<td>0.00</td>
<td>21.57</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>10.78</td>
<td>4.78</td>
<td>0.00</td>
<td>0.00</td>
<td>20.25</td>
<td></td>
</tr>
</tbody>
</table>
### RESPONSE SPECTRUM ANALYSIS

**MEMBER END FORCES**  
**STRUCTURE TYPE = PLANE**

---

**ALL UNITS ARE -- KNS METE (LOCAL)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>185.52</td>
<td>-23.47</td>
<td>0.00</td>
<td>0.00</td>
<td>-21.57</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-183.37</td>
<td>23.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-48.85</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>4.78</td>
<td>10.78</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>20.25</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>4.78</td>
<td>10.78</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>12.08</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>3</td>
<td>142.72</td>
<td>-9.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-1.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-141.11</td>
<td>9.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-45.70</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>135.56</td>
<td>-25.69</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-31.37</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-133.94</td>
<td>25.69</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-27.58</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>185.52</td>
<td>-23.47</td>
<td>0.00</td>
<td>0.00</td>
<td>21.57</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-183.37</td>
<td>-23.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>48.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>4.78</td>
<td>10.78</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>20.25</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4.78</td>
<td>10.78</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>12.08</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>142.72</td>
<td>25.69</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>27.58</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>135.56</td>
<td>9.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-133.94</td>
<td>25.69</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-27.58</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### RESPONSE SPECTRUM ANALYSIS

**MEMBER END FORCES**  
**STRUCTURE TYPE = PLANE**

---

**ALL UNITS ARE -- KNS METE (LOCAL)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>70.53</td>
<td>-43.63</td>
<td>0.00</td>
<td>0.00</td>
<td>-65.87</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-68.38</td>
<td>43.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-65.02</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>0.74</td>
<td>0.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.44</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.74</td>
<td>0.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2.20</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>53.45</td>
<td>-32.17</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-49.07</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-51.84</td>
<td>32.17</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-50.42</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>52.35</td>
<td>-33.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-49.73</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-50.74</td>
<td>33.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-47.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>70.53</td>
<td>43.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>65.87</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-68.38</td>
<td>43.63</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>65.02</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.74</td>
<td>0.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.44</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.74</td>
<td>0.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2.20</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>53.45</td>
<td>32.17</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>49.73</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-51.84</td>
<td>32.17</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>47.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>52.35</td>
<td>33.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>49.07</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-50.74</td>
<td>33.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>50.42</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>3</td>
<td>20.16</td>
<td>112.84</td>
<td>0.00</td>
<td>0.00</td>
<td>114.72</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>20.16</td>
<td>112.84</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-114.72</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

Application Examples

**EX. British Design Examples**

STAAD.Pro

User Manual
4 5 32.72 50.74 0.00 0.00 0.00 47.12
6 -32.72 51.84 0.00 0.00 0.00 -47.12

************* END OF LATEST ANALYSIS RESULT *************

59. LOAD LIST 1 3 4
60. PARAMETER
61. CODE EN 1993-1-1:2005
62. NA 1
63. SELECT ALL

STEEL DESIGN

STAAD.PRO MEMBER SELECTION - BS EN 1993-1-1:2005

NATIONAL ANNEX - NA to BS EN 1993-1-1:2005

PROGRAM CODE REVISION V1.13 BS_EC3_2005/1

RESPONSE SPECTRUM ANALYSIS -- PAGE NO. 10

ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)

MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
FX MY MZ LOCATION

=======================================================================
1 ST UC152X152X37 (BRITISH SECTIONS) PASS EC-6.3.3-662 0.901

1 183.37 C 0.00 -48.85 3.00
2 ST UC152X152X37 (BRITISH SECTIONS) PASS EC-6.3.3-662 0.901

1 183.37 C 0.00 48.85 3.00
3 ST UC152X152X44 (BRITISH SECTIONS) PASS EC-6.3.3-662 0.851

1 70.53 C 0.00 -65.87 0.00
4 ST UC152X152X44 (BRITISH SECTIONS) PASS EC-6.3.3-662 0.851

1 70.53 C 0.00 65.87 0.00
5 ST UB305X165X54 (BRITISH SECTIONS) PASS EC-6.3.2 LTB 0.980

1 20.16 T 0.00 114.72 0.00
6 ST UB305X165X40 (BRITISH SECTIONS) PASS EC-6.3.3-662 0.972

1 43.63 C 0.00 65.02 0.00

************* END OF TABULATED RESULT OF DESIGN *************

64. FINISH

RESPONSE SPECTRUM ANALYSIS -- PAGE NO. 11

************* END OF THE STAAD.Pro RUN *************

*****************************************************************************
**WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS. IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES. TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS. FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT COMMANDS.
*****************************************************************************

************* END OF THE STAAD.Pro RUN *************

*****************************************************************************************************
**** DATE= APR 14,2019  TIME= 22:52:48 ****
*****************************************************************************************************
* For technical assistance on STAAD.Pro, please visit *

Application Examples
EX. British Design Examples
EX. UK-12 Moving Load Generation on a Bridge Deck

This example demonstrates generation of load cases for the type of loading known as a moving load. This type of loading occurs classically when the load-causing units move on the structure, as in the case of trucks on a bridge deck. The mobile loads are discretized into several individual immobile load cases at discrete positions. During this process, enormous number of load cases may be created resulting in plenty of output to be sorted. To avoid looking into a lot of output, the maximum force envelope is requested for a few specific members.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-12 Moving Load Generation on a Bridge Deck.STD when you install the program.
Where:

\[ L_1 = 9 \text{ m} \]
\[ L_2 = 1.5 \text{ m} \]

Actual input is shown in bold lettering followed by explanation.

**STAAD FLOOR A SIMPLE BRIDGE DECK**

Every input has to start with the term STAAD. The word FLOOR signifies that the structure is a floor structure and the geometry is defined through X and Z axis.

**UNITS METER KNS**
Defines the input units for the data that follows.

**JOINT COORDINATES**

<table>
<thead>
<tr>
<th>Joint number followed by X, Y, and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). The first line generates joints 1 through 6. With the repeat (R) command, the coordinates of the next 30 joints are generated by repeating the pattern of the coordinates of the first 6 joints 5 times with X, Y and Z increments of 0, 0, and 9 respectively.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0 0 6 7.5 0 0</td>
</tr>
<tr>
<td>R 5 0 0 9.0</td>
</tr>
</tbody>
</table>

**MEMBER INCIDENCES**

<table>
<thead>
<tr>
<th>Defines the members by the joints to which they are connected. The fourth number indicates the final member number up to which they will be generated. Repeat all (abbreviated as R A) will create members by repeating the member incidence pattern of the previous 11 members. The number of repetitions to be carried out is provided after the RA command and the member increment and joint increment are defined as 11 and 6 respectively. The fifth line of input defines the member incidences for members 56 to 60.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 7 6</td>
</tr>
<tr>
<td>7 1 2 11</td>
</tr>
<tr>
<td>R A 4 11 6</td>
</tr>
<tr>
<td>56 31 32 60</td>
</tr>
</tbody>
</table>

**MEMBER PROPERTIES BRITISH**

<table>
<thead>
<tr>
<th>Properties for all members are assigned from the British steel table. The word ST stands for standard single section.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 60 TA ST UB305X165X40</td>
</tr>
</tbody>
</table>

**SUPPORTS**

<table>
<thead>
<tr>
<th>Pinned supports are specified at the above joints. A pinned support is one which can resist only translational forces.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 6 31 TO 36 PINNED</td>
</tr>
</tbody>
</table>

**UNITS MMS**

<table>
<thead>
<tr>
<th>The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. The units of length are changed from METER to MMS.</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNIT METER KNS</td>
</tr>
</tbody>
</table>

The characteristics of the vehicle are defined above in METER and KNS units. The above lines represent the first out of two sets of data required in moving load generation. The type number (1) is a label for identification of the load-causing unit, such as a truck. Three axles (90 90 45) are specified with the LOAD command. The spacing between the axles in the direction of movement (longitudinal direction) is specified after the DISTANCE
command. WIDTH is the spacing in the transverse direction, that is, it is the distance between the 2 prongs of an axle of the truck.

LOAD 1

Load case 1 is initiated.

SELF Y -1.0

Selfweight of the structure acting in the negative (due to the factor -1.0) global Y direction is the only component of load case 1.

LOAD GENERATION 10
TYPE 1 2.25 0. 0. ZI 3.0

This constitutes the second of the two sets of data required for moving load generation. 10 load cases are generated using the Type 1 vehicle whose characteristics were described earlier. For the first of these load cases, the X, Y and Z location of the reference load (see section 5.31.1 of the Technical Reference Manual (on page 2541)) have been specified after the command TYPE 1. The Z Increment of 3.0m denotes that the vehicle moves along the Z direction and the individual positions which are 3.0m apart will be used to generate the remaining 9 load cases.

As seen in Section 5.31.1 of the Technical Reference manual, the reference wheel is on the last axle. The first load case which is generated will be the one for which the first axle is just about to enter the bridge. The last load case should be the one for which the last axle is just about to exit the bridge. Thus, the total distance travelled by the reference load will be the length of the vehicle (distance from first axle to last axle) plus the span of the bridge. In this problem, that comes to

\[(3.0+1.5) + 45 = 49.5 \text{ m} \]

This example uses 3.0 m increments and generates 10 load cases.

However, if you want the vehicle to move forward in 4.5 m increments (each 4.5 m increment will create a discrete position of the truck on the bridge), it would required \((49.5/4.5)+1 = 12\) cases to be generated.

PERFORM ANALYSIS PRINT LOAD

The above command instructs the program to proceed with the analysis and print the values and positions of all the generated load cases.

PRINT MAXFORCE ENVELOP LIST 3 41 42

A maximum force envelope consisting of the highest forces for each degree of freedom on the listed members will be written into the output file.

FINISH

This command terminates the STAAD run.

Input File

STAAD FLOOR A SIMPLE BRIDGE DECK
UNITS METER KNS
JOINT COORDINATES
1 0.0 0.0 0.0 6 7.5 0.0 0.0
R 5 0.0 0.0 9.0
MEMBER INCIDENCES
1 1 7 6
7 1 2 11
R A 4 11 6
56 31 32 60
MEMBER PROPERTIES BRITISH
1 TO 60 TA ST UB305X165X40
SUPPORTS
1 TO 6 31 TO 36 PINNED
UNITS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER KNS
DEFINE MOVING LOAD
TYPE 1 LOAD 90.0 90.0 45.0 DISTANCE 3.0 1.5 WIDTH 3.0
LOAD 1
SELF Y -1.0
LOAD GENERATION 10
TYPE 1 2.25 0.0 0.0 ZI 3.0
PERFORM ANALYSIS PRINT LOAD
PRINT MAXFORCE ENVELOP LIST 3 41 42
FINISH

STAAD Output File

************************************************************************************
****************************** STAAD.Pro CONNECT Edition ******************************
************************ Version 22.01.00.** *********************************
************************ Proprietary Program of *****************************
************************ Bentley Systems, Inc. *****************************
************************ Date= APR 14, 2019 ***********************************
************************ Time= 22:52:52 *************************************
************************ Licensed to: Bentley Systems Inc ***********************
************************************************************************************

1. STAAD FLOOR A SIMPLE BRIDGE DECK

INPUT FILE: UK-12 Moving Load Generation on a Bridge Deck.STD

2. UNITS METER KNS
3. JOINT COORDINATES
4. 1 0.0 0.0 0.0 6 7.5 0.0 0.0
5. R 5 0.0 0.0 9.0
6. MEMBER INCIDENCES
7. 1 1 7 6
8. 7 1 2 11
9. R A 4 11 6
10. 56 31 32 60
11. MEMBER PROPERTIES BRITISH
12. 1 TO 60 TA ST UB305X165X40
13. SUPPORTS
14. 1 TO 6 31 TO 36 PINNED
15. UNITS MMS
16. DEFINE MATERIAL START
17. ISOTROPIC STEEL
18. E 210
19. POISSON 0.3
20. DENSITY 7.6977E-008
21. ALPHA 6E-006
22. DAMP 0.03
23. TYPE STEEL
24. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
25. END DEFINE MATERIAL
26. CONSTANTS
27. MATERIAL STEEL ALL
28. UNIT METER KNS
29. DEFINE MOVING LOAD
30. TYPE 1 LOAD 90.0 90.0 45.0 DISTANCE 3.0 1.5 WIDTH 3.0
31. LOAD 1
32. SELF Y -1.0
33. LOAD GENERATION 10
34. TYPE 1 2.25 0. 0. ZI 3.0
35. PERFORM ANALYSIS PRINT LOAD

A SIMPLE BRIDGE DECK

--- PROBLEM STATISTICS ---

NUMBER OF JOINTS 36  NUMBER OF MEMBERS 60
NUMBER OF PLATES 0  NUMBER OF SOLIDS 0
NUMBER OF SURFACES 0  NUMBER OF SUPPORTS 12

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 11, TOTAL DEGREES OF FREEDOM = 96
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

A SIMPLE BRIDGE DECK

LOADING 1

SELFWEIGHT Y -1.000
ACTUAL WEIGHT OF THE STRUCTURE = 124.391 KNS

LOADING 2

MEMBER LOAD - UNIT KNS METE
MEMBER UDL L1 L2 CON L LIN1 LIN2
8  -90.0000 GY 0.75
10  -90.0000 GY 0.75
3  -45.0000 GY 3.00
2  -45.0000 GY 3.00
5  -45.0000 GY 3.00
4  -45.0000 GY 3.00
3  -22.5000 GY 4.50
2  -22.5000 GY 4.50
5  -22.5000 GY 4.50
4  -22.5000 GY 4.50

LOADING 3

MEMBER LOAD - UNIT KNS METE
MEMBER UDL L1 L2 CON L LIN1 LIN2
3  -45.0000 GY 3.00
2  -45.0000 GY 3.00
5  -45.0000 GY 3.00
4  -45.0000 GY 3.00
3  -45.0000 GY 6.00
2  -45.0000 GY 6.00
### Member Load - Unit KNS

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>-45.0000</td>
<td>6.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-45.0000</td>
<td>6.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-22.5000</td>
<td>7.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-22.5000</td>
<td>7.50</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**LOADING** 4

---

### A SIMPLE BRIDGE DECK

---

**LOADING** 5

---

### A SIMPLE BRIDGE DECK

---

**LOADING** 6

---

### A SIMPLE BRIDGE DECK

---

**LOADING** 7

---

### Application Examples

EX. British Design Examples

---

**STAAD.Pro**

4741

User Manual
16  -45.0000 GY  6.00
15  -45.0000 GY  6.00
30  -90.0000 GY  0.75
32  -90.0000 GY  0.75
25  -22.5000 GY  1.50
24  -22.5000 GY  1.50
27  -22.5000 GY  1.50
26  -22.5000 GY  1.50

LOADING 8
---------
MEMBER LOAD - UNIT KNS  METE
MEMBER   UDL  L1 L2  CON   L  LIN1  LIN2
30  -90.0000 GY  0.75
32  -90.0000 GY  0.75
25  -45.0000 GY  3.00
24  -45.0000 GY  3.00
27  -45.0000 GY  3.00
26  -45.0000 GY  3.00
25  -22.5000 GY  4.50
24  -22.5000 GY  4.50
27  -22.5000 GY  4.50
26  -22.5000 GY  4.50

LOADING 9
---------
MEMBER LOAD - UNIT KNS  METE
MEMBER   UDL  L1 L2  CON   L  LIN1  LIN2
25  -45.0000 GY  3.00
24  -45.0000 GY  3.00
27  -45.0000 GY  3.00
26  -45.0000 GY  3.00

A SIMPLE BRIDGE DECK
-- PAGE NO. 6

LOADING 10
---------
MEMBER LOAD - UNIT KNS  METE
MEMBER   UDL  L1 L2  CON   L  LIN1  LIN2
25  -45.0000 GY  6.00
24  -45.0000 GY  6.00
27  -45.0000 GY  6.00
26  -45.0000 GY  6.00
25  -22.5000 GY  7.50
24  -22.5000 GY  7.50
27  -22.5000 GY  7.50
26  -22.5000 GY  7.50

LOADING 11
---------
MEMBER LOAD - UNIT KNS  METE
MEMBER   UDL  L1 L2  CON   L  LIN1  LIN2
41  -90.0000 GY  0.75
43  -90.0000 GY  0.75
36  -22.5000 GY  1.50
35  -22.5000 GY  1.50
38  -22.5000 GY  1.50
37  -22.5000 GY  1.50

LOADING 12
43                           -90.0000 GY   0.75
36                           -45.0000 GY   3.00
35                           -45.0000 GY   3.00
38                           -45.0000 GY   3.00
37                           -45.0000 GY   3.00
36                           -22.5000 GY   4.50
35                           -22.5000 GY   4.50
38                           -22.5000 GY   4.50
37                           -22.5000 GY   4.50

************ END OF DATA FROM INTERNAL STORAGE ************

36. PRINT MAXFORCE ENVELOP LIST 3 41 42
MAXFORCE ENVELOP LIST     3
A SIMPLE BRIDGE DECK -- PAGE NO.   7
MEMBER FORCE ENVELOPE
---------------------
ALL UNITS ARE KNS METE
MAX AND MIN FORCE VALUES AMONGST ALL SECTION LOCATIONS
MEMB          FY/    DIST  LD        MZ/    DIST  LD
FZ     DIST  LD        MY     DIST  LD        FX     DIST  LD
3 MAX     81.16   0.00    3       0.03   0.00    4
0.00   0.00    1       0.00   0.00    1 0.00   0.00    1
MIN -31.34   9.00    3     -504.76   9.00    5
0.00   9.00   11       0.00   9.00   11 0.00   9.00   11
41 MAX    73.48   0.00   10       9.18   1.50    5
0.00   0.00    1       0.00   0.00    1 0.00   0.00    1
MIN -18.36   1.50   11  -147.24   0.75   10
0.00   1.50   11       0.00   1.50   11 0.00   1.50   11
42 MAX   0.30   0.00    1       9.17   0.00    5
0.00   0.00    1       0.00   0.00    1 0.00   0.00    1
MIN -0.30   1.50    1  -134.84   1.50   10
0.00   1.50   11       0.00   1.50   11 0.00   1.50   11

*********** END OF FORCE ENVELOPE FROM INTERNAL STORAGE ***********

37. FINISH

Related Links
• TR.31.1 Definition of Moving Load System (on page 2541)
• V. Moving Load Generator (on page 3546)

EX. UK-13 Section Displacements for a Frame

Calculation of displacements at intermediate points of members of a plane frame is demonstrated in this example.
This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-13 Section Displacements for a Frame.STD when you install the program.

![Diagram of Example Problem No. 13](image)

Figure 496: Example Problem No. 13

Where:

- $L_1 = 4.5 \text{ m}$, $L_2 = 6.0 \text{ m}$
- $P = 25 \text{ kN}$
- $w = 45 \text{ kN/m}$

The dashed line represents the deflected shape of the structure. The shape is generated on the basis of displacements at the ends plus several intermediate points of the members.

Actual input is shown in bold lettering followed by explanation.

```plaintext
STAAD PLANE TEST FOR SECTION DISPLACEMENT

UNIT METER KNS

JOINT COORDINATES
1 0 0 ; 2 0.4.5 ; 3 6 4.5 ; 4 6 0.

MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
```

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

Application Examples
EX. British Design Examples

STAAD.Pro 4744 User Manual
Defines the members by the joints to which they are connected.

<table>
<thead>
<tr>
<th>MEMBER PROPERTY BRITISH</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 3 TABLE ST UC203X203X46</td>
</tr>
<tr>
<td>2 TABLE ST UB305X165X40</td>
</tr>
</tbody>
</table>

Member properties are specified from the British steel table. The word ST stands for standard single section.

<table>
<thead>
<tr>
<th>UNIT MMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEFINE MATERIAL START</td>
</tr>
<tr>
<td>ISOTROPIC STEEL</td>
</tr>
<tr>
<td>E 210</td>
</tr>
<tr>
<td>POISSON 0.3</td>
</tr>
<tr>
<td>DENSITY 7.6977e-008</td>
</tr>
<tr>
<td>ALPHA 6e-006</td>
</tr>
<tr>
<td>DAMP 0.03</td>
</tr>
<tr>
<td>TYPE STEEL</td>
</tr>
<tr>
<td>STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2</td>
</tr>
<tr>
<td>END DEFINE MATERIAL</td>
</tr>
<tr>
<td>CONSTANTS</td>
</tr>
<tr>
<td>MATERIAL STEEL ALL</td>
</tr>
</tbody>
</table>

The length unit is changed from METER to MMS. The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

<table>
<thead>
<tr>
<th>SUPPORT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 FIXED ; 4 PINNED</td>
</tr>
</tbody>
</table>

A fixed support is specified at Joint 1 and a pinned support at Joint 4.

<table>
<thead>
<tr>
<th>UNIT METER</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOADING 1 DEAD + LIVE + WIND</td>
</tr>
</tbody>
</table>

Load case 1 is initiated followed by an optional title.

<table>
<thead>
<tr>
<th>JOINT LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 FX 25.0</td>
</tr>
</tbody>
</table>

Load 1 contains a joint load of 25KN at node 2. FX indicates that the load is a force in the global X direction.

<table>
<thead>
<tr>
<th>MEMBER LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 UNI GY -45.0</td>
</tr>
</tbody>
</table>

Load 1 contains a member load also. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load.

<table>
<thead>
<tr>
<th>PERFORM ANALYSIS</th>
</tr>
</thead>
</table>

This command instructs the program to proceed with the analysis.

<table>
<thead>
<tr>
<th>PRINT MEMBER FORCES</th>
</tr>
</thead>
</table>

The above PRINT command is self-explanatory.

* FOLLOWS PRINT COMMAND WILL PRINT
  * DISPLACEMENTS OF THE MEMBERS
  * CONSIDERING EVERY TWELFTH INTERMEDIATE
  * POINT (THAT IS TOTAL OF 13 POINTS). THESE
  * DISPLACEMENTS ARE MEASURED IN GLOBAL X
  * Y Z COORDINATE SYSTEM AND THE VALUES
  * ARE FROM ORIGINAL COORDINATES (UNDEFLECTED
  * POSITION) OF CORRESPONDING TWELFTH
  * POINTS.
Above PRINT command is explained in the comment lines above.

This command terminates the STAAD run.

Input File

STAAD PLANE TEST FOR SECTION DISPLACEMENT
UNIT METER KNS
JOINT COORDINATES
1 0.0 0.0 ; 2 0.0 4.5 ; 3 6.0 4.5 ; 4 6.0 0.0
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
MEMBER PROPERTY BRITISH
1 3 TABLE ST UC203X203X46
2 TABLE ST UB305X165X40
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210.0
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORT
1 FIXED ; 4 PINNED
UNIT METER
LOADING 1 DEAD + LIVE + WIND
JOINT LOAD
2 FX 25.0
MEMBER LOAD
2 UNI GY -45.0
PERFORM ANALYSIS
PRINT MEMBER FORCES
*
* FOLLOWING PRINT COMMAND WILL PRINT
* DISPLACEMENTS OF THE MEMBERS CONSIDERING EVERY
* TWELFTH INTERMEDIATE POINT (THAT IS TOTAL OF 13 POINTS).
* THESE DISPLACEMENTS ARE MEASURED IN GLOBAL X Y Z
* COORDINATE SYSTEM AND THE VALUES ARE FROM
* ORIGINAL COORDINATES (UNDEFLECTED POSITION)
* OF THE CORRESPONDING TWELFTH POINTS.
MAX LOCAL DISPLACEMENT IS ALSO PRINTED. THE
LOCATION OF MAXIMUM INTERMEDIATE DISPLACEMENT
IS DETERMINED. THIS VALUE IS MEASURED FROM ABOVE
LOCATION TO THE STRAIGHT LINE JOINING START AND
END JOINTS OF THE DEFLECTED MEMBER.

PRINT SECTION DISPLACEMENT
FINISH

1. STAAD PLANE TEST FOR SECTION DISPLACEMENT
INPUT FILE: UK-13 Section Displacements for a Frame.STD
2. UNIT METER KNS
3. JOINT COORDINATES
   1 0.0 0.0 ; 2 0.0 4.5 ; 3 6.0 4.5 ; 4 6.0 0.0
5. MEMBER INCIDENCE
   1 1 2 ; 2 2 3 ; 3 3 4
7. MEMBER PROPERTY BRITISH
   1 3 TABLE ST UC203X203X46
   2 TABLE ST UB305X165X40
10. UNIT MMS
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
   E 210.0
14. POISSON 0.3
15. DENSITY 7.6977E-008
16. ALPHA 6E-006
17. DAMP 0.03
18. TYPE STEEL
19. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
20. END DEFINE MATERIAL
21. CONSTANTS
22. MATERIAL STEEL ALL
23. SUPPORT
24. 1 FIXED ; 4 PINNED
25. UNIT METER
26. LOADING 1 DEAD + LIVE + WIND
27. JOINT LOAD
28. 2 FX 25.0
29. MEMBER LOAD
30. 2 UNI GY -45.0
31. PERFORM ANALYSIS
   TEST FOR SECTION DISPLACEMENT
   -- PAGE NO. 2
   PROBLEM STATISTICS
   -----------------------------
NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       7
TOTAL LOAD COMBINATION CASES =     0 SO FAR.
32. PRINT MEMBER FORCES
MEMBER FORCES
TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    3
MEMBER END FORCES    STRUCTURE TYPE = PLANE
-----------------
ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
1    1     1    121.84      1.65     0.00      0.00      0.00      33.54
2   -121.84     -1.65     0.00      0.00      0.00     -26.10
2    1     2     23.35    121.84     0.00      0.00      0.00      26.10
3   -23.35    148.16     0.00      0.00      0.00    -105.06
3    1     3    148.16     23.35     0.00      0.00      0.00     105.06
4   -148.16    -23.35     0.00      0.00      0.00      -0.00
************** END OF LATEST ANALYSIS RESULT **************
33. *
34. * FOLLOWING PRINT COMMAND WILL PRINT
35. * DISPLACEMENTS OF THE MEMBERS CONSIDERING EVERY
36. * TWELFTH INTERMEDIATE POINT (THAT IS TOTAL OF 13 POINTS).
37. * THESE DISPLACEMENTS ARE MEASURED IN GLOBAL X Y Z
38. * COORDINATE SYSTEM AND THE VALUES ARE FROM
39. * ORIGINAL COORDINATES (UNDEFLECTED POSITION)
40. * OF THE CORRESPONDING TWELFTH POINTS.
41. *
42. * MAX LOCAL DISPLACEMENT IS ALSO PRINTED. THE
43. * LOCATION OF MAXIMUM INTERMEDIATE DISPLACEMENT
44. * IS DETERMINED. THIS VALUE IS MEASURED FROM ABOVE
45. * LOCATION TO THE STRAIGHT LINE JOINING START AND
46. * END JOINTS OF THE DEFLECTED MEMBER.
47. *
48. PRINT SECTION DISPLACEMENT
SECTION DISPLACE
TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    4
MEMBER SECTION DISPLACEMENTS
----------------------------
UNITS ARE - CM
MEMB  LOAD GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
1     1       0.0000    0.0000    0.0000    0.0249   -0.0037    0.0000
0.0981   -0.0074    0.0000    0.2187   -0.0111    0.0000
0.3856   -0.0148    0.0000    0.5981   -0.0185    0.0000
0.8552   -0.0222    0.0000    1.1560   -0.0260    0.0000
1.4996   -0.0297    0.0000    1.8850   -0.0334    0.0000
2.3115   -0.0445    0.0000    2.7780   -0.0408    0.0000
3.2836   -0.0445    0.0000    3.2836   -0.7367    0.0000
3.2814   -1.3907    0.0000    3.2824   -1.9447    0.0000
3.2793   -2.5329    0.0000    3.2782   -2.5850    0.0000
3.2771   -2.6266    0.0000    3.2760   -2.4791    0.0000
3.2749   -2.1595    0.0000    3.2739   -1.7007    0.0000
3.2728   -1.1514    0.0000    3.2717   -0.5758    0.0000
3.2706   -0.0541    0.0000
MAX LOCAL  DISP =    0.78662   AT     225.00  LOAD    1   L/DISP=    572
EX. UK-14 P-Delta Analysis of a Frame Under Seismic Loads

A space frame is analyzed for seismic loads. The seismic loads are generated using the procedures of the building code. A P-Delta analysis is performed to obtain the secondary effects of the lateral and vertical loads acting simultaneously.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-14 P-Delta Analysis of a Frame Under Seismic Loads.STD when you install the program.
STAAD SPACE EXAMPLE PROBLEM FOR UBC LOAD

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame.

UNIT METER KNS

Defines the input units for the data that follows.

JOINT COORDINATES
1 0 0 4 10.5 0 0
REPEAT 3 0 0 3.5
REPEAT ALL 3 0 3.5 0

The X, Y and Z coordinates of the joints are specified here. First, coordinates of joints 1 through 4 are generated by taking advantage of the fact that they are equally spaced. Then, this pattern is REPEATED 3 times with a Z increment of 3.5 m for each repetition to generate joints 5 to 16. The REPEAT ALL command will then repeat 3 times, the pattern of joints 1 to 16 to generate joints 17 to 64.

MEMBER INCIDENCES
* beams in x direction
101 17 18 103
104 21 22 106
107 25 26 109

Figure 497: Example Problem No. 14
Defines the members by the joints to which they are connected. Following the specification of incidences for members 101 to 112, the REPEAT ALL command is used to repeat the pattern and generate incidences for members 113 through 136. A similar logic is used in specification of incidences of members 201 through 212 and generation of incidences for members 213 to 236. Finally, members incidences of columns 301 to 348 are specified.

The beam members have prismatic member property specification (YD & ZD) while the columns (members 301 to 348) have their properties called from the built-in British steel table.

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

**Tip:** You may see these values with the help of the command PRINT MATERIAL PROPERTIES following the preceding commands.
Indicates the joints where the supports are located as well as the type of support restraints.

```
DEFINE UBC LOAD
ZONE 0.2 I 1.0 RwX 9 RwZ 9 S 1.5 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7.0
49 TO 64 WEIGHT 3.5
```

There are two stages in a static seismic load. The first stage is to define the code-specified load parameters along with the vertical loads (weights) from which the base shear will be calculated. The vertical loads may be specified in the form of selfweight, joint weights and/or member weights. Member weights are not shown in this example. It is important to note that these vertical loads are used purely in the determination of the horizontal base shear only. In other words, the structure is not analyzed for these vertical loads.

```
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -7.0
49 TO 64 FY -3.5
```

This is the second stage in which the static seismic load is applied with the help of a load case number, corresponding direction (X in the above case) and a factor by which the generated horizontal loads should be multiplied. Along with the seismic lateral load, deadweight is also added to the same load case. Since we will be doing second-order (PDELTA) analysis, it is important that we include horizontal and vertical loads in the same load case.

```
LOAD 2
UBC LOAD Z 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -7.0
49 TO 64 FY -3.5
```

In load case 2, the static seismic load is being applied in the Z direction. Vertical loads are part of this case, also.

```
PDELTA ANALYSIS PRINT LOAD DATA
```

We are requesting a second-order analysis by specifying the command `PDELTA ANALYSIS. PRINT LOAD DATA` is used to obtain a report of all the applied and generated loadings.

```
PRINT SUPPORT REACTIONS
FINISH
```

The above commands are self-explanatory.

**Input File**

```
STAAD SPACE EXAMPLE PROBLEM FOR UBC LOAD
UNIT METER KNS
JOINT COORDINATES
1 0 0 4 10.5 0 0
REPEAT 3 0 3.5
REPEAT ALL 3 0 3.5 0
MEMBER INCIDENCES
* beams in x direction
101 17 18 103
104 21 22 106
107 25 26 109
```
110 29 30 112
REPEAT ALL 2 12 16
* beams in z direction
201 17 21 204
205 21 25 208
209 25 29 212
REPEAT ALL 2 12 16
* columns
301 1 17 348
MEMBER PROPERTIES BRITISH
101 TO 136 201 TO 236 PRIS YD 0.40 ZD 0.30
301 TO 348 TA ST UB457X152X52
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 301 TO 348
MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
UNIT METER
SUPPORT
1 TO 16 FIXED
DEFINE UBC LOAD
ZONE 0.2 I 1.0 RNX 9 RWZ 9 S 1.5 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7.0
49 TO 64 WEIGHT 3.5
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOAD
17 TO 48 FY -7.0
49 TO 64 FY -3.5
LOAD 2
UBC LOAD Z 0.75
SELFWEIGHT Y -1.0
JOINT LOAD
17 TO 48 FY -7.0
49 TO 64 FY -3.5
PDELTA ANALYSIS PRINT LOAD DATA
## STAAD Output File

**INPUT FILE: UK-14 P-Delta Analysis of a Frame Under Seismic Loads.**

### 1. STAAD SPACE EXAMPLE PROBLEM FOR UBC LOAD

**UNIT METER KNS**

**JOINT COORDINATES**

1. 0 0 0 4 10.5 0 0
2. REPEAT 3 0 0 3.5
3. REPEAT ALL 3 0 3.5 0

**MEMBER INCIDENCES**

8. * BEAMS IN X DIRECTION
9. 101 17 18 103
10. 104 21 22 106
11. 107 25 26 109
12. 110 29 30 112
13. REPEAT ALL 2 12 16
14. * BEAMS IN Z DIRECTION
15. 201 17 21 204
16. 205 21 25 208
17. 209 25 29 212
18. REPEAT ALL 2 12 16
19. * COLUMNS
20. 301 1 17 348
21. MEMBER PROPERTIES BRITISH
22. 101 TO 136 201 TO 236 PRIS YD 0.40 ZD 0.30
23. 301 TO 348 TA ST UB457X152X52
24. UNIT MMS
25. DEFINE MATERIAL START
26. ISOTROPIC STEEL
27. E 210
28. POISSON 0.3
29. DENSITY 7.6977E-008
30. ALPHA 6E-006
31. DAMP 0.03
32. TYPE STEEL
33. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
34. ISOTROPIC CONCRETE
35. E 21.0
36. POISSON 0.17
37. DENSITY 2.36158E-008
38. ALPHA 5E-006
39. DAMP 0.05
40. G 9.25
41. TYPE CONCRETE
42. STRENGTH FCU 0.0275
43. END DEFINE MATERIAL
44. CONSTANTS
45. MATERIAL STEEL MEMB 301 TO 348
46. MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
47. UNIT METER
48. SUPPORT
49. 1 TO 16 FIXED
50. DEFINE UBC LOAD
51. ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
52. SELFWEIGHT
53. JOINT WEIGHT
54. 17 TO 48 WEIGHT 7.0
55. 49 TO 64 WEIGHT 3.5
56. LOAD 1
57. UBC LOAD X 0.75
58. SELFWEIGHT Y -1.0
59. JOINT LOAD
60. 17 TO 48 FY -7.0
61. 49 TO 64 FY -3.5
62. LOAD 2
63. UBC LOAD Z 0.75
64. SELFWEIGHT Y -1.0
65. JOINT LOAD
66. 17 TO 48 FY -7.0
67. 49 TO 64 FY -3.5
68. PDELTA ANALYSIS PRINT LOAD DATA

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS         64  NUMBER OF MEMBERS     120
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     288
TOTAL LOAD COMBINATION CASES =     0 SO FAR.
EXAMPLE PROBLEM FOR UBC LOAD -- PAGE NO. 3
EXAMPLE PROBLEM FOR UBC LOAD -- PAGE NO. 4
LOADING 1

*******************************
SELFWEIGHT Y -1.000
ACTUAL WEIGHT OF THE STRUCTURE =  800.269 KNS

LOADING - UNIT KNS METE

JOINT FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM-Z
17  0.00 -7.00  0.00  0.00  0.00  0.00
18  0.00 -7.00  0.00  0.00  0.00  0.00
19  0.00 -7.00  0.00  0.00  0.00  0.00
20  0.00 -7.00  0.00  0.00  0.00  0.00
21  0.00 -7.00  0.00  0.00  0.00  0.00
22  0.00 -7.00  0.00  0.00  0.00  0.00
23  0.00 -7.00  0.00  0.00  0.00  0.00
24  0.00 -7.00  0.00  0.00  0.00  0.00
25  0.00 -7.00  0.00  0.00  0.00  0.00
26  0.00 -7.00  0.00  0.00  0.00  0.00
27  0.00 -7.00  0.00  0.00  0.00  0.00
28  0.00 -7.00  0.00  0.00  0.00  0.00
EXAMPLE PROBLEM FOR UBC LOAD

LOADING

SELFWEIGHT Y -1.000

ACTUAL WEIGHT OF THE STRUCTURE = 800.269 KNS

JOINT LOAD - UNIT KNS METE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>18</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>21</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>22</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>23</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>24</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>25</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>26</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>27</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>28</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>29</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>30</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>31</td>
<td>0.00</td>
<td>-7.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

STAAD.Pro 4756 User Manual
EXAMPLE PROBLEM FOR UBC LOAD

**WARNING: IF THIS UBC/IBC ANALYSIS HAS TENSION/COMPRESSION OR REPEAT LOAD OR RE-ANALYSIS OR SELECT OPTIMIZE, THEN EACH UBC/IBC CASE SHOULD BE FOLLOWED BY PERFORM ANALYSIS & CHANGE.**

X DIRECTION : Ta = 0.455 Tb = 0.285 Tuser = 0.000
C = 2.7500, LOAD FACTOR = 0.750
UBC TYPE = 94
UBC FACTOR V = 0.0611 x 1080.27 = 66.02 KNS

Z DIRECTION : Ta = 0.455 Tb = 1.092 Tuser = 0.000
C = 2.7500, LOAD FACTOR = 0.750
UBC TYPE = 94
UBC FACTOR V = 0.0611 x 1080.27 = 66.02 KNS

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LATERAL LOAD (KNS)</th>
<th>TORSIONAL LOAD (KNS -METE)</th>
<th>FACTOR</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>FX 0.449</td>
<td>MY 0.000</td>
<td>0.750</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>FX 0.569</td>
<td>MY 0.000</td>
<td>0.750</td>
<td></td>
</tr>
<tr>
<td>Joint</td>
<td>FZ</td>
<td>MY</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>-----</td>
<td>-----</td>
<td>-------</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>0.449</td>
<td>MY</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
<td></td>
</tr>
</tbody>
</table>

### EXAMPLE PROBLEM FOR UBC LOAD

---

<table>
<thead>
<tr>
<th>Joint</th>
<th>FX</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>20</td>
<td>0.449</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>21</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>22</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>23</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>24</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>25</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>26</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>27</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>28</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>29</td>
<td>0.449</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

TOTAL = 9.097 0.000 AT LEVEL 3.500 METE

<table>
<thead>
<tr>
<th>Joint</th>
<th>FX</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>31</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>32</td>
<td>0.449</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

TOTAL = 18.194 0.000 AT LEVEL 7.000 METE

<table>
<thead>
<tr>
<th>Joint</th>
<th>FX</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>49</td>
<td>1.032</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>50</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>51</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>52</td>
<td>1.032</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>53</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>54</td>
<td>1.746</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>55</td>
<td>1.746</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>56</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>57</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>58</td>
<td>1.746</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>59</td>
<td>1.746</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>60</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>61</td>
<td>1.032</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>62</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>63</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>64</td>
<td>1.032</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

TOTAL = 22.222 0.000 AT LEVEL 10.500 METE

---

### JOINT LATERAL TORSIONAL LOAD - 2 LOAD (KNS ) MOMENT (KNS -METE) FACTOR - 0.750

<table>
<thead>
<tr>
<th>Joint</th>
<th>FZ</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>0.449</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>18</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

---

STAAD.Pro 4758 User Manual
**Application Examples**

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>FZ</th>
<th>0.569</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>FZ</td>
<td>0.449</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

**EXAMPLE PROBLEM FOR UBC LOAD**

---

**Page No. 8**

<table>
<thead>
<tr>
<th>FZ</th>
<th>0.569</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>FZ</td>
<td>0.449</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>0.569</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>0.688</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

---

**TOTAL =**

9.097 0.000 AT LEVEL 3.500 METE

<table>
<thead>
<tr>
<th>FZ</th>
<th>0.899</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>FZ</td>
<td>1.137</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>1.137</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>0.899</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>1.137</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

---

**TOTAL =**

18.194 0.000 AT LEVEL 7.000 METE

<table>
<thead>
<tr>
<th>FZ</th>
<th>1.032</th>
<th>MY</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>FZ</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>1.032</td>
<td>MY</td>
<td>0.000</td>
</tr>
<tr>
<td>FZ</td>
<td>1.389</td>
<td>MY</td>
<td>0.000</td>
</tr>
</tbody>
</table>

---

**TOTAL =**

22.222 0.000 AT LEVEL 10.500 METE

++ Adjusting Displacements.

******* END OF DATA FROM INTERNAL STORAGE *******

69. PRINT SUPPORT REACTIONS

SUPPORT REACTION

**STAAD.Pro 4759 User Manual**
### EXAMPLE PROBLEM FOR UBC LOAD

**SUPPORT REACTIONS - UNIT KNS**

**STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-2.21</td>
<td>43.45</td>
<td>0.06</td>
<td>-0.07</td>
<td>-0.00</td>
<td>5.81</td>
</tr>
<tr>
<td>2</td>
<td>0.46</td>
<td>41.46</td>
<td>-3.04</td>
<td>-5.63</td>
<td>0.00</td>
<td>-0.50</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-3.40</td>
<td>65.65</td>
<td>0.06</td>
<td>0.07</td>
<td>-0.00</td>
<td>7.09</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>56.21</td>
<td>-3.01</td>
<td>-5.63</td>
<td>0.00</td>
<td>-0.01</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-3.43</td>
<td>64.48</td>
<td>0.06</td>
<td>0.07</td>
<td>-0.00</td>
<td>7.13</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.01</td>
<td>56.21</td>
<td>-3.01</td>
<td>-5.63</td>
<td>0.00</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-3.13</td>
<td>57.17</td>
<td>0.06</td>
<td>0.07</td>
<td>-0.00</td>
<td>6.80</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.46</td>
<td>41.46</td>
<td>-3.04</td>
<td>-5.63</td>
<td>0.00</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-2.31</td>
<td>62.90</td>
<td>-0.01</td>
<td>-0.01</td>
<td>-0.00</td>
<td>6.02</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.46</td>
<td>73.97</td>
<td>-3.14</td>
<td>-5.82</td>
<td>0.00</td>
<td>-0.50</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-3.52</td>
<td>85.32</td>
<td>-0.01</td>
<td>-0.01</td>
<td>-0.00</td>
<td>7.33</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>88.73</td>
<td>-3.11</td>
<td>-5.83</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-3.54</td>
<td>84.12</td>
<td>-0.01</td>
<td>-0.01</td>
<td>-0.00</td>
<td>7.37</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.01</td>
<td>88.73</td>
<td>-3.11</td>
<td>-5.83</td>
<td>0.00</td>
<td>0.02</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-3.22</td>
<td>77.05</td>
<td>-0.01</td>
<td>-0.01</td>
<td>-0.00</td>
<td>7.02</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.46</td>
<td>73.97</td>
<td>-3.14</td>
<td>-5.82</td>
<td>0.00</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-2.31</td>
<td>62.90</td>
<td>0.01</td>
<td>0.01</td>
<td>0.00</td>
<td>6.02</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.46</td>
<td>65.97</td>
<td>-3.14</td>
<td>-5.81</td>
<td>0.00</td>
<td>-0.50</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-3.52</td>
<td>85.32</td>
<td>0.01</td>
<td>0.01</td>
<td>0.00</td>
<td>7.33</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>80.72</td>
<td>-3.11</td>
<td>-5.81</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-3.54</td>
<td>84.12</td>
<td>0.01</td>
<td>0.01</td>
<td>0.00</td>
<td>7.37</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.01</td>
<td>80.72</td>
<td>-3.11</td>
<td>-5.81</td>
<td>0.00</td>
<td>0.02</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-3.22</td>
<td>77.05</td>
<td>0.01</td>
<td>0.01</td>
<td>0.00</td>
<td>7.02</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.46</td>
<td>65.97</td>
<td>-3.14</td>
<td>-5.81</td>
<td>0.00</td>
<td>0.50</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-2.21</td>
<td>43.45</td>
<td>-0.06</td>
<td>-0.07</td>
<td>0.00</td>
<td>5.81</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.46</td>
<td>59.15</td>
<td>-3.12</td>
<td>-5.76</td>
<td>0.00</td>
<td>-0.50</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>-3.40</td>
<td>65.65</td>
<td>-0.06</td>
<td>-0.07</td>
<td>0.00</td>
<td>7.09</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.01</td>
<td>73.91</td>
<td>-3.09</td>
<td>-5.76</td>
<td>0.00</td>
<td>-0.02</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>-3.43</td>
<td>64.48</td>
<td>-0.06</td>
<td>-0.07</td>
<td>0.00</td>
<td>7.13</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.01</td>
<td>73.91</td>
<td>-3.09</td>
<td>-5.76</td>
<td>0.00</td>
<td>0.02</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-3.13</td>
<td>57.17</td>
<td>-0.06</td>
<td>-0.07</td>
<td>0.00</td>
<td>6.80</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.46</td>
<td>59.15</td>
<td>-3.12</td>
<td>-5.76</td>
<td>0.00</td>
<td>0.50</td>
<td></td>
</tr>
</tbody>
</table>

************** END OF LATEST ANALYSIS RESULT **************

**70. FINISH**

**EX. UK-15 Wind and Floor Load Generation on a Space Frame**

A space frame is analyzed for loads generated using the built-in wind and floor load generation facilities.
This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-15 Wind and Floor Load Generation on a Space Frame.STD when you install the program.

STAAD SPACE - WIND AND FLOOR LOAD GENERATION

This is a space frame analysis problem. Every STAAD input has to start with the command STAAD. The SPACE specification is used to denote a space (3D) frame.

UNIT METER KNS

The UNIT specification is used to specify the length and/or force units to be used.

JOINT COORDINATES
1 0 0 0
2 4 0 0
3 9 0 0
4 0 0 4
5 4 0 4
6 0 0 8
7 4 0 8
8 9 0 8
REPEAT ALL 2 0 3.5 0
The JOINT COORDINATE specification is used to specify the X, Y, and Z coordinates of the joints. Note that the REPEAT ALL command has been used to generate joints for the two upper stories each with a Y increment of 3.5 m.

MEMBER INCIDENCES
* Columns
  1 1 9 16
* Beams in the X direction
  17 9 10 18
  19 12 13
  20 14 15 21
  22 17 18 23
  24 20 21
  25 22 23 26
* Beams in the Z direction
  27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
  32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24

The MEMBER INCIDENCE specification is used for specifying member connectivities.

MEMBER PROPERTIES BRITISH
1 TO 16 TA ST UB457X191X74
17 TO 26 TA ST UB457X152X52
27 TO 36 TA ST UB457X152X52

Properties for all members are specified from the built-in BRITISH steel table. Three different sections have been used.

UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
  E 210
  POISSON 0.3
  DENSITY 7.6977e-008
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

SUPPORT
1 TO 8 FIXED BUT MX MZ

The supports of the structure are defined through the SUPPORT specification. Here all the supports are FIXED with releases specified in the MX (rotation about global X-axis) and MZ (rotation about global Z-axis) directions.

DEFINE WIND LOAD
TYPE 1
  INTENSITY 1.0 1.5 HEIGHT 3.5 7.0
  EXPOSURE 0.90 YRANGE 6.0 8.0
  EXPOSURE 0.85 JOINT 9 12 14

When a structure has to be analyzed for wind loading, the engineer is confronted with the task of first converting an abstract quantity like wind velocity or wind pressure into concentrated loads at joints, distributed loads on members, or pressure loads on plates. The large number of calculations involved in this conversion can be
avoided by making use of STAAD's wind load generation utility. This utility takes wind pressure at various
heights as the input, and converts them to values that can then be used as concentrated forces known as joint
loads in specific load cases. The input specification is done in two stages. The first stage is initiated above
through the DEFINE WIND LOAD command. The basic parameters of the WIND loading are specified here. All
values need to be provided in the current UNIT system. Each wind category is identified with a TYPE number (an
identification mark) which is used later to specify load cases.

In this example, two different wind intensities (1.0 KN/sq. m and 1.5 KN/sq. m) are specified for two different
height zones (0 to 3.5 m and 3.5 to 7.0 m). The EXPOSURE specification is used to mitigate or magnify the effect at
specific nodes due to special considerations like openings in the structure. In this case, two different exposure
factors are specified. The first EXPOSURE specification specifies the exposure factor as 0.9 for all joints within
the height range (defined as global Y-range) of 6.0m – 8.0m. The second EXPOSURE specification specifies the
exposure factor as 0.85 for joints 9, 12 and 14. In the EXPOSURE factor specification, the joints may be specified
directly or through a vertical range specification.

LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1

This is the second stage of input specification for the wind load generation. The term WIND LOAD and the
direction term that follows are used to specify the wind loading in a particular lateral direction. In this case,
WIND loading TYPE 1, defined previously, is being applied in the global X-direction with a positive
multiplication factor of 1.2.

LOAD 2 FLOOR LOAD @ Y = 3.5M AND 7M
FLOOR LOAD
YRANGE 3.4 3.6 FLOAD -5.0 XRANGE 0.0 4.0 ZRANGE 0.0 8.0
YRANGE 3.4 3.6 FLOAD -2.5 XRANGE 4.0 9.0 ZRANGE 0.0 8.0
YRANGE 6.9 7.1 FLOAD -2.5

In load case 2 in this problem, a floor load generation is performed. In a floor load generation, a pressure load
(force per unit area) is converted by the program into specific points forces and distributed forces on the
members located in that region. The YRANGE, XRANGE, and ZRANGE specifications are used to define the area of
the structure on which the pressure is acting. The FLOAD specification is used to specify the value of that
pressure. All values need to be provided in the current UNIT system. For example, in the first line in the above
FLOOR LOAD specification, the region is defined as being located within the bounds YRANGE of 3.4 – 3.6 m,
XRANGE of 0.0 - 4.0 m and ZRANGE of 0.0 - 8.0 m. The -5.0 signifies that the pressure is 5.0 KN/sq.m. in the
negative global Y direction.

The program will identify the members lying within the specified region and derive member loads on these
members based on two-way load distribution.

PERFORM ANALYSIS PRINT LOAD DATA

We can view the values and position of the generated loads with the help of the PRINT LOAD DATA command
used above along with the PERFORM ANALYSIS command.

PRINT SUPPORT REACTION
FINISH

Above commands are self-explanatory.

Input File

STAAD SPACE - WIND AND FLOOR LOAD GENERATION
UNIT METER KNS
JOINT COORDINATES
1 0 0 0
2 4 0 0
3 9 0 0
4 0 0 4
5 4 0 4
6 0 0 8
7 4 0 8
8 9 0 8
REPEAT ALL 2 0 3.5 0
MEMBER INCIDENCES
* Columns
1 1 9 16
* Beams in the X direction
17 9 10 18
19 12 13
20 14 15 21
22 17 18 23
24 20 21
25 22 23 26
* Beams in the Z direction
27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
MEMBER PROPERTIES BRITISH
1 TO 16 TA ST UB457X191X74
17 TO 26 TA ST UB457X152X52
27 TO 36 TA ST UB457X152X52
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
SUPPORT
1 TO 8 FIXED BUT MX MZ
DEFINE WIND LOAD
TYPE 1
INTENSITY 1.0 1.5 HEIGHT 3.5 7.0
EXPOSURE 0.90 YRANGE 6.0 8.0
EXPOSURE 0.85 JOINT 9 12 14
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
LOAD 2 FLOOR LOAD AT Y = 3.5M AND 7M
FLOOR LOAD
YRANGE 3.4 3.6 FLOAD -5.0 XRANGE 0.0 4.0 ZRANGE 0.0 8.0
YRANGE 3.4 3.6 FLOAD -2.5 XRANGE 4.0 9.0 ZRANGE 0.0 8.0
YRANGE 6.9 7.1 FLOAD -2.5
PERFORM ANALYSIS PRINT LOAD DATA
PRINT SUPPORT REACTION
FINISH
1. STAAD SPACE - WIND AND FLOOR LOAD GENERATION

INPUT FILE: UK-15 Wind and Floor Load Generation on a Space Frame.STD

2. UNIT METER KNS
3. JOINT COORDINATES
   1 0 0 0
   2 4 0 0
   3 9 0 0
   4 0 0 4
   5 4 0 4
   6 0 0 8
   7 4 0 8
   8 9 0 8
12. REPEAT ALL 2 0 3.5 0
13. MEMBER INCIDENCES
14. * COLUMNS
15. 1 1 9 16
16. * BEAMS IN THE X DIRECTION
17. 17 9 10 18
18. 19 12 13
19. 20 14 15 21
20. 22 17 18 23
21. 24 20 21
22. 25 22 23 26
23. * BEAMS IN THE Z DIRECTION
24. 27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
25. 32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
26. MEMBER PROPERTIES BRITISH
27. 1 TO 16 TA ST UB457X191X74
28. 17 TO 26 TA ST UB457X152X52
29. 27 TO 36 TA ST UB457X152X52
30. UNIT MMS
31. DEFINE MATERIAL START
32. ISOTROPIC STEEL
33. E 210
34. POISSON 0.3
35. DENSITY 7.6977E-008
36. ALPHA 6E-006
37. DAMP 0.03
38. TYPE STEEL
39. WIND AND FLOOR LOAD GENERATION
40. END DEFINE MATERIAL
41. CONSTANTS
42. MATERIAL STEEL ALL
43. UNIT METER
44. SUPPORT
45. 1 TO 8 FIXED BUT MX MZ
46. DEFINE WIND LOAD

*** NOTE: If any floor diaphragm is present in the model Wind Load definition should be defined after Floor Diaphragm definition. Otherwise wind load generation may be unsuccessful during analysis.

47. TYPE 1
48. INTENSITY 1.0 1.5 HEIGHT 3.5 7.0
49. EXPOSURE 0.90 YRANGE 6.0 8.0
50. EXPOSURE 0.85 JOINT 9 12 14
51. LOAD 1 WIND LOAD IN X-DIRECTION
52. WIND LOAD X 1.2 TYPE 1
53. LOAD 2 FLOOR LOAD AT Y = 3.5M AND 7M
54. FLOOR LOAD
55. YRANGE 3.4 3.6 LOAD -5.0 X RANGE 0.0 4.0 Z RANGE 0.0 8.0

**NOTE** about Floor/OneWay Loads/Weights.
Please note that depending on the shape of the floor you may have to break up the FLOOR/ONEWAY LOAD into multiple commands.
For details please refer to Technical Reference Manual Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.

56. YRANGE 3.4 3.6 LOAD -2.5 X RANGE 4.0 9.0 Z RANGE 0.0 8.0
57. YRANGE 6.9 7.1 LOAD -2.5

58. PERFORM ANALYSIS PRINT LOAD DATA

** PROBLEM STATISTICS **
-----------------------------------
NUMBER OF JOINTS         24  NUMBER OF MEMBERS      36
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      8

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     112
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
- WIND AND FLOOR LOAD GENERATION -- PAGE NO. 3
- WIND AND FLOOR LOAD GENERATION -- PAGE NO. 4

LOADING 1 WIND LOAD IN X-DIRECTION

JOINT LOAD - UNIT KNS METE

<table>
<thead>
<tr>
<th>JOINT</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>8.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>4.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>8.93</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>12</td>
<td>17.85</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>14</td>
<td>8.93</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>17</td>
<td>5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>20</td>
<td>11.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>22</td>
<td>5.67</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

LOADING 2 FLOOR LOAD AT Y = 3.5M AND 7M

MEMBER LOAD - UNIT KNS METE

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>UDL</th>
<th>L1</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>-0.1563</td>
<td>GY</td>
<td>0.17</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.4688</td>
<td>GY</td>
<td>0.39</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.7812</td>
<td>GY</td>
<td>0.63</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.0938</td>
<td>GY</td>
<td>0.88</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.4062</td>
<td>GY</td>
<td>1.13</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>Story</th>
<th>X-Position</th>
<th>Y-Position</th>
<th>Wind Load</th>
<th>Floor Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>-1.7188 GY</td>
<td>1.38</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-2.0312 GY</td>
<td>1.63</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-2.3438 GY</td>
<td>1.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-2.3438 GY</td>
<td>2.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-2.0312 GY</td>
<td>2.37</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.7188 GY</td>
<td>2.62</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.4062 GY</td>
<td>2.87</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-1.0938 GY</td>
<td>3.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.7812 GY</td>
<td>3.37</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.4688 GY</td>
<td>3.61</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-0.1563 GY</td>
<td>3.83</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.1563 GY</td>
<td>0.17</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.4688 GY</td>
<td>0.39</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-0.7812 GY</td>
<td>0.63</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.0938 GY</td>
<td>0.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.4062 GY</td>
<td>1.13</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.7188 GY</td>
<td>1.38</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-2.0312 GY</td>
<td>1.63</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-2.3438 GY</td>
<td>1.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-2.3438 GY</td>
<td>2.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-2.0312 GY</td>
<td>2.37</td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>-1.7188 GY</td>
<td>2.62</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**WIND AND FLOOR LOAD GENERATION**

---

-- PAGE NO. 5
### Application Examples

EX. British Design Examples

<table>
<thead>
<tr>
<th>Floor</th>
<th>X-Direction Load</th>
<th>Y-Direction Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>27</td>
<td>-0.4688 GY</td>
<td>3.61</td>
</tr>
<tr>
<td>27</td>
<td>-0.1563 GY</td>
<td>3.83</td>
</tr>
<tr>
<td>19</td>
<td>-0.1563 GY</td>
<td>0.17</td>
</tr>
<tr>
<td>19</td>
<td>-0.4688 GY</td>
<td>0.39</td>
</tr>
<tr>
<td>19</td>
<td>-0.7812 GY</td>
<td>0.63</td>
</tr>
<tr>
<td>19</td>
<td>-1.0938 GY</td>
<td>0.88</td>
</tr>
<tr>
<td>19</td>
<td>-1.4062 GY</td>
<td>1.13</td>
</tr>
<tr>
<td>19</td>
<td>-1.7188 GY</td>
<td>1.38</td>
</tr>
<tr>
<td>19</td>
<td>-2.0312 GY</td>
<td>1.63</td>
</tr>
<tr>
<td>19</td>
<td>-2.3438 GY</td>
<td>1.88</td>
</tr>
<tr>
<td>19</td>
<td>-2.3438 GY</td>
<td>2.12</td>
</tr>
<tr>
<td>19</td>
<td>-2.0312 GY</td>
<td>2.37</td>
</tr>
<tr>
<td>19</td>
<td>-1.7188 GY</td>
<td>2.62</td>
</tr>
<tr>
<td>19</td>
<td>-1.4062 GY</td>
<td>2.87</td>
</tr>
<tr>
<td>19</td>
<td>-1.0938 GY</td>
<td>3.12</td>
</tr>
<tr>
<td>19</td>
<td>-0.7812 GY</td>
<td>3.37</td>
</tr>
<tr>
<td>19</td>
<td>-0.4688 GY</td>
<td>3.61</td>
</tr>
<tr>
<td>19</td>
<td>-0.1563 GY</td>
<td>3.83</td>
</tr>
<tr>
<td>20</td>
<td>-0.1563 GY</td>
<td>0.17</td>
</tr>
<tr>
<td>20</td>
<td>-0.4688 GY</td>
<td>0.39</td>
</tr>
<tr>
<td>20</td>
<td>-0.7812 GY</td>
<td>0.63</td>
</tr>
<tr>
<td>20</td>
<td>-1.0938 GY</td>
<td>0.88</td>
</tr>
<tr>
<td>20</td>
<td>-1.4062 GY</td>
<td>1.13</td>
</tr>
<tr>
<td>20</td>
<td>-1.7188 GY</td>
<td>1.38</td>
</tr>
<tr>
<td>20</td>
<td>-2.0312 GY</td>
<td>1.63</td>
</tr>
<tr>
<td>20</td>
<td>-2.3438 GY</td>
<td>1.88</td>
</tr>
<tr>
<td>20</td>
<td>-2.3438 GY</td>
<td>2.12</td>
</tr>
<tr>
<td>20</td>
<td>-2.0312 GY</td>
<td>2.37</td>
</tr>
<tr>
<td>20</td>
<td>-1.7188 GY</td>
<td>2.62</td>
</tr>
<tr>
<td>20</td>
<td>-1.4062 GY</td>
<td>2.87</td>
</tr>
<tr>
<td>20</td>
<td>-1.0938 GY</td>
<td>3.12</td>
</tr>
<tr>
<td>20</td>
<td>-0.7812 GY</td>
<td>3.37</td>
</tr>
<tr>
<td>20</td>
<td>-0.4688 GY</td>
<td>3.61</td>
</tr>
<tr>
<td>20</td>
<td>-0.1563 GY</td>
<td>3.83</td>
</tr>
<tr>
<td>28</td>
<td>-0.1563 GY</td>
<td>0.17</td>
</tr>
<tr>
<td>28</td>
<td>-0.4688 GY</td>
<td>0.39</td>
</tr>
<tr>
<td>28</td>
<td>-0.7812 GY</td>
<td>0.63</td>
</tr>
<tr>
<td>28</td>
<td>-1.0938 GY</td>
<td>0.88</td>
</tr>
<tr>
<td>28</td>
<td>-1.4062 GY</td>
<td>1.13</td>
</tr>
<tr>
<td>28</td>
<td>-1.7188 GY</td>
<td>1.38</td>
</tr>
<tr>
<td>28</td>
<td>-2.0312 GY</td>
<td>1.63</td>
</tr>
</tbody>
</table>

- WIND AND FLOOR LOAD GENERATION

-- PAGE NO. 6

STAAD.Pro 4768 User Manual
## Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD (Gy)</th>
<th>UNIT</th>
<th>LOAD (KN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>28</td>
<td>-2.3438</td>
<td>GY</td>
<td>1.88</td>
</tr>
<tr>
<td>28</td>
<td>-2.3438</td>
<td>GY</td>
<td>2.12</td>
</tr>
<tr>
<td>28</td>
<td>-2.0312</td>
<td>GY</td>
<td>2.37</td>
</tr>
<tr>
<td>28</td>
<td>-1.7188</td>
<td>GY</td>
<td>2.62</td>
</tr>
<tr>
<td>28</td>
<td>-1.4062</td>
<td>GY</td>
<td>2.87</td>
</tr>
<tr>
<td>28</td>
<td>-1.0938</td>
<td>GY</td>
<td>3.12</td>
</tr>
<tr>
<td>28</td>
<td>-0.7812</td>
<td>GY</td>
<td>3.37</td>
</tr>
<tr>
<td>28</td>
<td>-0.4688</td>
<td>GY</td>
<td>3.61</td>
</tr>
<tr>
<td>28</td>
<td>-0.1563</td>
<td>GY</td>
<td>3.83</td>
</tr>
</tbody>
</table>

### MEMBER LOAD - UNIT KNS METE

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>UDL</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>18</td>
<td>-0.1221</td>
<td>GY</td>
<td>0.21</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.3662</td>
<td>GY</td>
<td>0.49</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.6184</td>
<td>GY</td>
<td>0.79</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-0.8545</td>
<td>GY</td>
<td>1.10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>-1.0986</td>
<td>GY</td>
<td>1.41</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### WIND AND FLOOR LOAD GENERATION

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD (Gy)</th>
<th>UNIT</th>
<th>LOAD (KN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>18</td>
<td>-1.3428</td>
<td>GY</td>
<td>1.72</td>
</tr>
<tr>
<td>18</td>
<td>-1.5869</td>
<td>GY</td>
<td>2.04</td>
</tr>
<tr>
<td>18</td>
<td>-1.8311</td>
<td>GY</td>
<td>2.35</td>
</tr>
<tr>
<td>18</td>
<td>-1.8311</td>
<td>GY</td>
<td>2.65</td>
</tr>
<tr>
<td>18</td>
<td>-1.5869</td>
<td>GY</td>
<td>2.96</td>
</tr>
<tr>
<td>18</td>
<td>-1.3428</td>
<td>GY</td>
<td>3.28</td>
</tr>
<tr>
<td>18</td>
<td>-1.0986</td>
<td>GY</td>
<td>3.59</td>
</tr>
<tr>
<td>18</td>
<td>-0.8545</td>
<td>GY</td>
<td>3.90</td>
</tr>
<tr>
<td>18</td>
<td>-0.6184</td>
<td>GY</td>
<td>4.21</td>
</tr>
<tr>
<td>18</td>
<td>-0.3662</td>
<td>GY</td>
<td>4.51</td>
</tr>
<tr>
<td>18</td>
<td>-0.1221</td>
<td>GY</td>
<td>4.79</td>
</tr>
<tr>
<td>31</td>
<td>-0.1221</td>
<td>GY</td>
<td>0.21</td>
</tr>
<tr>
<td>31</td>
<td>-0.3662</td>
<td>GY</td>
<td>0.49</td>
</tr>
<tr>
<td>31</td>
<td>-0.6184</td>
<td>GY</td>
<td>0.79</td>
</tr>
<tr>
<td>31</td>
<td>-0.8545</td>
<td>GY</td>
<td>1.10</td>
</tr>
<tr>
<td>31</td>
<td>-1.0986</td>
<td>GY</td>
<td>1.41</td>
</tr>
<tr>
<td>31</td>
<td>-1.3428</td>
<td>GY</td>
<td>1.72</td>
</tr>
<tr>
<td>31</td>
<td>-1.5869</td>
<td>GY</td>
<td>2.04</td>
</tr>
<tr>
<td>31</td>
<td>-1.8311</td>
<td>GY</td>
<td>2.35</td>
</tr>
<tr>
<td>31</td>
<td>-6.2500</td>
<td>GY</td>
<td>2.50</td>
</tr>
<tr>
<td>31</td>
<td>-1.8311</td>
<td>GY</td>
<td>5.50</td>
</tr>
<tr>
<td>31</td>
<td>-1.5869</td>
<td>GY</td>
<td>5.96</td>
</tr>
<tr>
<td>31</td>
<td>-1.3428</td>
<td>GY</td>
<td>6.28</td>
</tr>
<tr>
<td>31</td>
<td>-1.0986</td>
<td>GY</td>
<td>6.59</td>
</tr>
<tr>
<td>31</td>
<td>-0.8545</td>
<td>GY</td>
<td>6.90</td>
</tr>
<tr>
<td>31</td>
<td>-0.6184</td>
<td>GY</td>
<td>7.21</td>
</tr>
<tr>
<td>31</td>
<td>-0.3662</td>
<td>GY</td>
<td>7.51</td>
</tr>
<tr>
<td>31</td>
<td>-0.1221</td>
<td>GY</td>
<td>7.79</td>
</tr>
<tr>
<td>21</td>
<td>-0.1221</td>
<td>GY</td>
<td>0.21</td>
</tr>
<tr>
<td>21</td>
<td>-0.3662</td>
<td>GY</td>
<td>0.49</td>
</tr>
<tr>
<td>21</td>
<td>-0.6184</td>
<td>GY</td>
<td>0.79</td>
</tr>
<tr>
<td>21</td>
<td>-0.8545</td>
<td>GY</td>
<td>1.10</td>
</tr>
<tr>
<td>21</td>
<td>-1.0986</td>
<td>GY</td>
<td>1.41</td>
</tr>
<tr>
<td>21</td>
<td>-1.3428</td>
<td>GY</td>
<td>1.72</td>
</tr>
<tr>
<td>21</td>
<td>-1.5869</td>
<td>GY</td>
<td>2.04</td>
</tr>
<tr>
<td>21</td>
<td>-1.8311</td>
<td>GY</td>
<td>2.35</td>
</tr>
<tr>
<td>21</td>
<td>-1.8311</td>
<td>GY</td>
<td>2.65</td>
</tr>
<tr>
<td>21</td>
<td>-1.5869</td>
<td>GY</td>
<td>2.96</td>
</tr>
<tr>
<td>21</td>
<td>-1.3428</td>
<td>GY</td>
<td>3.28</td>
</tr>
<tr>
<td>21</td>
<td>-1.0986</td>
<td>GY</td>
<td>3.59</td>
</tr>
<tr>
<td>21</td>
<td>-0.8545</td>
<td>GY</td>
<td>3.90</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. British Design Examples

<table>
<thead>
<tr>
<th>Member</th>
<th>UDL (kN/m)</th>
<th>L1</th>
<th>L2</th>
<th>CON</th>
<th>L (m)</th>
<th>LIN1</th>
<th>LIN2</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
<td>-0.0781</td>
<td>0.17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.2344</td>
<td>0.39</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.3906</td>
<td>0.63</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.5469</td>
<td>0.88</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.7031</td>
<td>1.13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.8594</td>
<td>1.38</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-1.0156</td>
<td>1.63</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-1.1719</td>
<td>1.88</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-1.1719</td>
<td>2.12</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-1.0156</td>
<td>2.37</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.8594</td>
<td>2.62</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.7031</td>
<td>2.87</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.5469</td>
<td>3.12</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.3906</td>
<td>3.37</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.2344</td>
<td>3.61</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>-0.0781</td>
<td>3.83</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.0781</td>
<td>0.17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.2344</td>
<td>0.39</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.3906</td>
<td>0.63</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.5469</td>
<td>0.88</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.7031</td>
<td>1.13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.8594</td>
<td>1.38</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-1.0156</td>
<td>1.63</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-1.1719</td>
<td>1.88</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-1.1719</td>
<td>2.12</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-1.0156</td>
<td>2.37</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.8594</td>
<td>2.62</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.7031</td>
<td>2.87</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.5469</td>
<td>3.12</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.3906</td>
<td>3.37</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.2344</td>
<td>3.61</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>-0.0781</td>
<td>3.83</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0781</td>
<td>0.17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.2344</td>
<td>0.39</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- WIND AND FLOOR LOAD GENERATION -- PAGE NO. 8

-6.2500 GY 0.00 1.50
-0.1221 GY 0.21
-0.3662 GY 0.49
-0.6104 GY 0.79
-0.8545 GY 1.10
-1.0986 GY 1.41
-1.3428 GY 1.72
-1.5869 GY 2.04
-1.8311 GY 2.35
-6.2500 GY 2.50 4.00

Member Load - Unit KNS METE
<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>24</td>
<td>-0.3906 GY</td>
<td>0.63</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.5469 GY</td>
<td>0.88</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.7031 GY</td>
<td>1.13</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.8594 GY</td>
<td>1.38</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-1.0156 GY</td>
<td>1.63</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-1.1719 GY</td>
<td>1.88</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-1.1719 GY</td>
<td>2.12</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-1.0156 GY</td>
<td>2.37</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.8594 GY</td>
<td>2.62</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.7031 GY</td>
<td>2.87</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.5469 GY</td>
<td>3.12</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.3906 GY</td>
<td>3.37</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.2344 GY</td>
<td>3.61</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>-0.0781 GY</td>
<td>0.17</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.2344 GY</td>
<td>0.39</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.3906 GY</td>
<td>0.63</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.5469 GY</td>
<td>0.88</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.7031 GY</td>
<td>1.13</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.8594 GY</td>
<td>1.38</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.0156 GY</td>
<td>1.63</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.1719 GY</td>
<td>1.88</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.1719 GY</td>
<td>2.12</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.0156 GY</td>
<td>2.37</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.8594 GY</td>
<td>2.62</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.7031 GY</td>
<td>2.87</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.5469 GY</td>
<td>3.12</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.3906 GY</td>
<td>3.37</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.2344 GY</td>
<td>3.61</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.0781 GY</td>
<td>0.17</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.1221 GY</td>
<td>0.21</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.3662 GY</td>
<td>0.49</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.6104 GY</td>
<td>0.79</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.8545 GY</td>
<td>1.10</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.0986 GY</td>
<td>1.41</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.3428 GY</td>
<td>1.72</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.5869 GY</td>
<td>2.04</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.8311 GY</td>
<td>2.35</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.8311 GY</td>
<td>2.65</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.5869 GY</td>
<td>2.96</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.3428 GY</td>
<td>3.28</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-1.0986 GY</td>
<td>3.59</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.8545 GY</td>
<td>3.90</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.6104 GY</td>
<td>4.21</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.3662 GY</td>
<td>4.51</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>-0.1221 GY</td>
<td>4.79</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.1221 GY</td>
<td>0.21</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.3662 GY</td>
<td>0.49</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.6104 GY</td>
<td>0.79</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-0.8545 GY</td>
<td>1.10</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-1.0986 GY</td>
<td>1.41</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-1.3428 GY</td>
<td>1.72</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-1.5869 GY</td>
<td>2.04</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-1.8311 GY</td>
<td>2.35</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-6.2500 GY</td>
<td>2.50</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-1.8311 GY</td>
<td>5.65</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>-1.5869 GY</td>
<td>5.96</td>
<td></td>
</tr>
</tbody>
</table>
### WIND AND FLOOR LOAD GENERATION

<table>
<thead>
<tr>
<th>Level</th>
<th>Wind Load (kN)</th>
<th>Floor Load (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>36</td>
<td>-1.3428 GY</td>
<td>6.28</td>
</tr>
<tr>
<td>36</td>
<td>-1.0986 GY</td>
<td>6.59</td>
</tr>
<tr>
<td>36</td>
<td>-0.8545 GY</td>
<td>6.90</td>
</tr>
<tr>
<td>36</td>
<td>-0.6104 GY</td>
<td>7.21</td>
</tr>
<tr>
<td>36</td>
<td>-0.3662 GY</td>
<td>7.51</td>
</tr>
<tr>
<td>36</td>
<td>-0.1221 GY</td>
<td>7.79</td>
</tr>
<tr>
<td>26</td>
<td>-0.1221 GY</td>
<td>0.21</td>
</tr>
<tr>
<td>26</td>
<td>-0.3662 GY</td>
<td>0.49</td>
</tr>
<tr>
<td>26</td>
<td>-0.6104 GY</td>
<td>0.79</td>
</tr>
</tbody>
</table>

- WIND AND FLOOR LOAD GENERATION

<table>
<thead>
<tr>
<th>Level</th>
<th>Wind Load (kN)</th>
<th>Floor Load (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>26</td>
<td>-0.8545 GY</td>
<td>1.10</td>
</tr>
<tr>
<td>26</td>
<td>-1.0986 GY</td>
<td>1.41</td>
</tr>
<tr>
<td>26</td>
<td>-1.3428 GY</td>
<td>1.72</td>
</tr>
<tr>
<td>26</td>
<td>-1.5869 GY</td>
<td>2.04</td>
</tr>
<tr>
<td>26</td>
<td>-1.8311 GY</td>
<td>2.35</td>
</tr>
<tr>
<td>26</td>
<td>-1.8311 GY</td>
<td>2.65</td>
</tr>
<tr>
<td>26</td>
<td>-1.5869 GY</td>
<td>2.96</td>
</tr>
<tr>
<td>26</td>
<td>-1.3428 GY</td>
<td>3.28</td>
</tr>
<tr>
<td>26</td>
<td>-1.0986 GY</td>
<td>3.59</td>
</tr>
<tr>
<td>26</td>
<td>-0.6104 GY</td>
<td>3.90</td>
</tr>
<tr>
<td>24</td>
<td>-0.3662 GY</td>
<td>4.21</td>
</tr>
<tr>
<td>24</td>
<td>-0.0781 GY</td>
<td>4.51</td>
</tr>
<tr>
<td>24</td>
<td>-0.1221 GY</td>
<td>4.79</td>
</tr>
<tr>
<td>26</td>
<td>-0.1221 GY</td>
<td>0.21</td>
</tr>
<tr>
<td>26</td>
<td>-0.3662 GY</td>
<td>0.49</td>
</tr>
<tr>
<td>26</td>
<td>-0.6104 GY</td>
<td>0.79</td>
</tr>
<tr>
<td>26</td>
<td>-0.8545 GY</td>
<td>1.10</td>
</tr>
<tr>
<td>26</td>
<td>-1.0986 GY</td>
<td>1.41</td>
</tr>
<tr>
<td>26</td>
<td>-1.3428 GY</td>
<td>1.72</td>
</tr>
<tr>
<td>26</td>
<td>-1.5869 GY</td>
<td>2.04</td>
</tr>
<tr>
<td>26</td>
<td>-1.8311 GY</td>
<td>2.35</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Level</th>
<th>Wind Load (kN)</th>
<th>Floor Load (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>35</td>
<td>-0.8545 GY</td>
<td>2.90</td>
</tr>
<tr>
<td>35</td>
<td>-0.6104 GY</td>
<td>3.21</td>
</tr>
<tr>
<td>35</td>
<td>-0.3662 GY</td>
<td>3.51</td>
</tr>
<tr>
<td>35</td>
<td>-0.1221 GY</td>
<td>3.79</td>
</tr>
<tr>
<td>35</td>
<td>-0.1221 GY</td>
<td>0.21</td>
</tr>
</tbody>
</table>

- WIND AND FLOOR LOAD GENERATION

<table>
<thead>
<tr>
<th>Level</th>
<th>Wind Load (kN)</th>
<th>Floor Load (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>35</td>
<td>-0.1221 GY</td>
<td>0.21</td>
</tr>
<tr>
<td>34</td>
<td>-0.3662 GY</td>
<td>0.49</td>
</tr>
<tr>
<td>34</td>
<td>-0.6104 GY</td>
<td>0.79</td>
</tr>
<tr>
<td>34</td>
<td>-0.8545 GY</td>
<td>1.10</td>
</tr>
<tr>
<td>34</td>
<td>-1.0986 GY</td>
<td>1.41</td>
</tr>
<tr>
<td>34</td>
<td>-1.3428 GY</td>
<td>1.72</td>
</tr>
<tr>
<td>34</td>
<td>-1.5869 GY</td>
<td>2.04</td>
</tr>
<tr>
<td>34</td>
<td>-1.8311 GY</td>
<td>2.35</td>
</tr>
<tr>
<td>34</td>
<td>-6.2500 GY</td>
<td>2.50</td>
</tr>
<tr>
<td>34</td>
<td>4.00</td>
<td>4.50</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Level</th>
<th>Wind Load (kN)</th>
<th>Floor Load (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>34</td>
<td>-6.2500 GY</td>
<td>0.00</td>
</tr>
<tr>
<td>34</td>
<td>0.50</td>
<td>1.50</td>
</tr>
<tr>
<td>35</td>
<td>-0.0781 GY</td>
<td>0.17</td>
</tr>
<tr>
<td>34</td>
<td>-0.0781 GY</td>
<td>0.39</td>
</tr>
<tr>
<td>34</td>
<td>-0.3996 GY</td>
<td>0.63</td>
</tr>
<tr>
<td>34</td>
<td>-0.5469 GY</td>
<td>0.88</td>
</tr>
<tr>
<td>34</td>
<td>-0.7031 GY</td>
<td>1.13</td>
</tr>
<tr>
<td>34</td>
<td>-0.8594 GY</td>
<td>1.38</td>
</tr>
<tr>
<td>34</td>
<td>-1.0156 GY</td>
<td>1.63</td>
</tr>
<tr>
<td>34</td>
<td>-1.1719 GY</td>
<td>1.88</td>
</tr>
<tr>
<td>34</td>
<td>-1.1719 GY</td>
<td>2.12</td>
</tr>
<tr>
<td>34</td>
<td>-1.1719 GY</td>
<td>2.37</td>
</tr>
<tr>
<td>34</td>
<td>-1.1719 GY</td>
<td>2.62</td>
</tr>
<tr>
<td>34</td>
<td>-1.1719 GY</td>
<td>2.87</td>
</tr>
<tr>
<td>34</td>
<td>-0.0781 GY</td>
<td>3.12</td>
</tr>
<tr>
<td>34</td>
<td>-0.0781 GY</td>
<td>3.37</td>
</tr>
<tr>
<td>34</td>
<td>-0.0781 GY</td>
<td>3.61</td>
</tr>
<tr>
<td>34</td>
<td>-0.0781 GY</td>
<td>3.83</td>
</tr>
<tr>
<td>35</td>
<td>-0.0781 GY</td>
<td>0.17</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
<td>---------</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>-9.58</td>
</tr>
<tr>
<td>2</td>
<td>0.80</td>
<td>25.69</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-6.70</td>
</tr>
</tbody>
</table>
EX. UK-16 Time History Analysis for Forcing Function and Ground Motion

Dynamic Analysis (Time History) is performed for a 3 span beam with concentrated and distributed masses. The structure is subjected to “forcing function” and “ground motion” loading. The maxima of joint displacements, member end forces and support reactions are determined.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-16 Time History Analysis for Forcing Function and Ground Motion.STD when you install the program.
Figure 499: Example Problem No. 16

Where:

L1 = 1.2 m

STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS

Every input file has to start with the word STAAD. The term PLANE signifies that the structure is a plane frame.

UNITS CMS KNS

Specifies the units to be used.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1  0.0  0.0  0.0</td>
</tr>
<tr>
<td>2  0.0 120.0  0.0</td>
</tr>
<tr>
<td>3  0.0 240.0  0.0</td>
</tr>
<tr>
<td>4  0.0 360.0  0.0</td>
</tr>
</tbody>
</table>

Joint number followed by the X, Y and Z coordinates are specified above.

<table>
<thead>
<tr>
<th>MEMBER INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 3</td>
</tr>
</tbody>
</table>

Incidences of members 1 to 3 are specified above.

<table>
<thead>
<tr>
<th>MEMBER PROPERTIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 2 3 PRIS AX 100.0 IZ 833.33</td>
</tr>
</tbody>
</table>
All the members have PRISMATIC property specification. Since this is a plane frame, Area of cross section (AX) and Moment of Inertia (IZ) about the Z axis are adequate for the analysis.

SUPPORTS 1 4 PINNED

Pinned supports are located at nodes 1 and 4.

DEFINE MATERIAL ISOTROPIC CONCRETE
E 2850
POISSON 0.17
DENSITY 25e-006
ALPHA 5e-006
DAMP 0.05
G 925
TYPE CONCRETE
STRENGTH FCU 2.75
END DEFINE MATERIAL

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

The following are the constants used:

UNIT NEWTON METER
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -20.0 0.5 100.0 1.0 200.0 1.5 500.0 2.0 800.0 2.5 500.0 3.0 70.0
TYPE 2 ACCELERATION
0.0 0.1 0.5 -0.25 1.0 -0.5 1.5 -0.9 2.0 -1.3 2.5 -1.0 3.0 -0.7
ARRIVAL TIMES
0.0
DAMPING 0.075

There are two stages in the command specification required for a time history analysis. The first stage is defined above. First the characteristics of the time varying load are provided. The loading type may be a forcing function (vibrating machinery) or ground motion (earthquake). The former is input in the form of time-force pairs while the latter is in the form of time-acceleration pairs. Following this data, all possible arrival times for these loads on the structure as well as the modal damping ratio are specified. In this example, the damping ratio is the same (7.5%) for all modes.

LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 500.0

Load case 1 above is a static load. A uniformly distributed force of 500 Newton/m acts along the global X direction on all 3 members.

LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOINT LOAD
2 3 FX 4000.0
TIME LOAD
2 3 FX 1 1
GROUND MOTION X 2 1

This is the second stage in the command specification for time history analysis. This involves the application of the time varying load on the structure. The masses that constitute the mass matrix of the structure are specified.
through the selfweight and joint load commands. The program will extract the lumped masses from these weights. Following that, both the TIME LOAD and GROUND MOTION are applied simultaneously.

**Note:** This example is only for illustration purposes and that it may be unlikely that a TIME LOAD and GROUND MOTION both act on the structure at the same time.

The Time load command is used to apply the Type 1 force, acting in the global X direction, at arrival time number 1, at nodes 2 and 3. The Ground motion, namely, the Type 2 time history loading, is also in the global X direction at arrival time 1.

PERFORM ANALYSIS

The above command initiates the analysis process.

PRINT JOINT DISPLACEMENTS

During the analysis, the program calculates joint displacements for every time step. The absolute maximum value of the displacement for every joint is then extracted from this joint displacement history. So, the value printed using the above command is the absolute maximum value for each of the six degrees of freedom at each node.

UNIT KNS METER
PRINT MEMBER FORCES
PRINT SUPPORT REACTION

The member forces and support reactions too are calculated for every time step. For each degree of freedom, the maximum value of the member force and support reaction is extracted from these histories and reported in the output file using the above command.

FINISH

**Input File**

STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
UNIT CMS KNS
JOINT COORDINATES
1 0.0 0.0 0.0
2 0.0 120.0 0.0
3 0.0 240.0 0.0
4 0.0 360.0 0.0
MEMBER INCIDENCES
1 1 2 3
MEMBER PROPERTIES
1 2 3 PRIS AX 100.0 IZ 833.33
SUPPORTS
1 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2850
POISSON 0.17
DENSITY 25e-006
ALPHA 5e-006
DAMP 0.05
G 925
TYPE CONCRETE
STRENGTH FCU 2.75
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT NEWTON METER
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -20.0 0.5 100.0 1.0 200.0 1.5 500.0 2.0 800.0 2.5 500.0 3.0 70.0
TYPE 2 ACCELERATION
0.0 0.1 0.5 -0.25 1.0 -0.5 1.5 -0.9 2.0 -1.3 2.5 -1.0 3.0 -0.7
ARRIVAL TIMES
0.0
DAMPING 0.075
LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 500.0
LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOINT LOAD
2 3 FX 4000.0
TIME LOAD
2 3 FX 1 1
GROUND MOTION X 2 1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
UNIT KNS METER
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
FINISH

STAAD Output File

************************************************************************************************* PAGE NO. 1
*                                                   *
*          STAAD.Pro CONNECT Edition                 *
*          Version 22.01.00.**                       *
*          Proprietary Program of                    *
*          Bentley Systems, Inc.                     *
*          Date= APR 14, 2019                        *
*          Time= 22:53:13                            *
*                                                   *
*      Licensed to: Bentley Systems Inc              *
*************************************************************************************************

1. STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
INPUT FILE: UK-16 Time History Analysis for Forcing Function and Ground Motion.STD
2. UNITS CMS KNS
3. JOINT COORDINATES
4. 1 0.0 0.0 0.0
5. 2 0.0 120.0 0.0
6. 3 0.0 240.0 0.0
7. 4 0.0 360.0 0.0
8. MEMBER INCIDENCES
9. 1 1 2 3
10. MEMBER PROPERTIES
11. 1 2 3 PRIS AX 100.0 IZ 833.33
12. SUPPORTS
13. 1 4 PINNED
14. DEFINE MATERIAL START
15. ISOTROPIC CONCRETE
16. E 2850
17. POISSON 0.17
18. DENSITY 25E-006
19. ALPHA 5E-006
20. DAMP 0.05
21. G 925
22. TYPE CONCRETE
23. STRENGTH FCU 2.75
24. END DEFINE MATERIAL
25. CONSTANTS
26. MATERIAL CONCRETE ALL
27. UNIT NEWTON METER
28. DEFINE TIME HISTORY
29. TYPE 1 FORCE
30. 0.0 -20.0 0.5 100.0 1.0 200.0 1.5 500.0 2.0 800.0 2.5 500.0 3.0 70.0
31. TYPE 2 ACCELERATION
32. 0.0 0.1 0.5 -0.25 1.0 -0.25 1.5 -0.9 2.0 -1.3 2.5 -1.0 3.0 -0.7
33. ARRIVAL TIMES
34. 0.0
35. DAMPING 0.075
36. LOAD 1 STATIC LOAD
37. MEMBER LOAD
38. 1 2 3 UNI GX 500.0
39. EXAMPLE FOR TIME HISTORY ANALYSIS
-- PAGE NO. 2
40. LOAD 2 TIME HISTORY LOAD
41. SELFWEIGHT X 1.0
42. SELFWEIGHT Y 1.0
43. JOINT LOAD
44. 2 3 FX 4000.0
45. TIME LOAD
46. 2 3 FX 1 1
47. GROUND MOTION X 2 1
48. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       8
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
***NOTE: MASSES DEFINED UNDER LOAD#       2 WILL FORM
THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
MORE MODES WERE REQUESTED THAN THERE ARE FREE MASSES.
NUMBER OF MODES REQUESTED              =     6
NUMBER OF EXISTING MASSES IN THE MODEL =     4
NUMBER OF MODES THAT WILL BE USED      =     4
***  EIGENSOLUTION : ADVANCED METHOD ***
EXAMPLE FOR TIME HISTORY ANALYSIS
-- PAGE NO. 3

CALCULATED FREQUENCIES FOR LOAD CASE

MODE FREQUENCY(CYCLES/SEC) PERIOD(SEC)
1 3.087 0.32397
2 11.955 0.08365
3 443.457 0.00226
4 768.090 0.00130

MODAL WEIGHT (MODAL MASS TIMES g) IN NEWT GENERALIZED
MODE X Y Z WEIGHT

Application Examples
EX. British Design Examples

STAAD.Pro 4779 User Manual
MASS PARTICIPATION FACTORS

Mass Participation Factors in Percent

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>100.00</td>
<td>0.00</td>
<td>0.00</td>
<td>100.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>100.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>100.00</td>
<td>0.00</td>
<td>100.000</td>
<td>100.000</td>
<td>0.000</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>100.000</td>
<td>100.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

Actual Modal Damping Used in Analysis

<table>
<thead>
<tr>
<th>MODE</th>
<th>DAMPING</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0750000</td>
</tr>
<tr>
<td>2</td>
<td>0.0750000</td>
</tr>
<tr>
<td>3</td>
<td>0.0750000</td>
</tr>
<tr>
<td>4</td>
<td>0.0750000</td>
</tr>
</tbody>
</table>

Time Step Used in Time History Analysis = 0.00139 Seconds

Number of Modes Whose Contribution Is Considered = 2

Example for Time History Analysis -- Page No. 4

WARNING: Number of Modes Limited to a Frequency of 360.0 Due to the DT Value Entered.

Time Duration of Time History Analysis = 3.000 Seconds

Number of Time Steps in the Solution Process = 2160

48. Print Joint Displacements

Base Shear Units Are -- Newt Mete

Maximum Base Shear X = -2.777266E+03 Y = 0.000000E+00 Z = 0.000000E+00

At Times

2.054167 0.000000 0.000000

Joint Displacement (CM Radians) Structure Type = Plane

49. Unit KNS Meter

50. Print Member Forces

Member Forces

Example for Time History Analysis -- Page No. 6

Member End Forces Structure Type = Plane

All Units Are -- KNS Mete (Local)
### EX. UK-17 User-Provided Tables

The usage of user-provided steel tables is illustrated in this example for the analysis and design of a plane frame. User-provided tables allow you to specify property data for sections not found in the built-in steel section tables.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-17 User-Provided Tables.STD when you install the program.
Figure 500: Example Problem No. 17

Where:

- \( L_1 = 4.5 \text{ m}, L_2 = 6.0 \text{ m} \)
- \( P = 25 \text{ kN} \)
- \( w = 45 \text{ kN/m} \)

Actual input is shown in bold lettering followed by explanation.

STAAD PLANE EXAMPLE FOR USER TABLE

Every input file has to start with the command STAAD. The PLANE command is used to designate the structure as a plane frame.

UNIT METER KNS
The UNIT command sets the length and force units to be used.

JOINT COORDINATES

<table>
<thead>
<tr>
<th>Joint</th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>2.9</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>3.0</td>
<td>6.0</td>
</tr>
<tr>
<td>4</td>
<td>6.9</td>
<td>6.0</td>
</tr>
<tr>
<td>5</td>
<td>7.0</td>
<td>10.5</td>
</tr>
<tr>
<td>6</td>
<td>9.0</td>
<td>10.5</td>
</tr>
<tr>
<td>7</td>
<td>9.2</td>
<td>10.5</td>
</tr>
<tr>
<td>8</td>
<td>10.6</td>
<td>7.5</td>
</tr>
<tr>
<td>9</td>
<td>11.4</td>
<td>7.5</td>
</tr>
<tr>
<td>10</td>
<td>13.2</td>
<td>7.5</td>
</tr>
</tbody>
</table>

The above set of data is used to provide joint coordinates for the various joints of the structure. The Cartesian system is being used here. The data consists of the joint number followed by global X and Y coordinates. Note that for a space frame, the Z coordinate(s) need to be provided also.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCES

<table>
<thead>
<tr>
<th>Member</th>
<th>Joint 1</th>
<th>Joint 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>7</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>6</td>
</tr>
<tr>
<td>4</td>
<td>6</td>
<td>8</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>5</td>
</tr>
<tr>
<td>7</td>
<td>10</td>
<td>16</td>
</tr>
<tr>
<td>8</td>
<td>11</td>
<td>15</td>
</tr>
<tr>
<td>9</td>
<td>12</td>
<td>13</td>
</tr>
<tr>
<td>10</td>
<td>14</td>
<td>15</td>
</tr>
<tr>
<td>11</td>
<td>15</td>
<td>16</td>
</tr>
<tr>
<td>12</td>
<td>13</td>
<td>14</td>
</tr>
<tr>
<td>13</td>
<td>16</td>
<td>11</td>
</tr>
<tr>
<td>14</td>
<td>17</td>
<td>11</td>
</tr>
<tr>
<td>15</td>
<td>18</td>
<td>10</td>
</tr>
<tr>
<td>16</td>
<td>19</td>
<td>10</td>
</tr>
<tr>
<td>17</td>
<td>20</td>
<td>7</td>
</tr>
<tr>
<td>18</td>
<td>21</td>
<td>9</td>
</tr>
<tr>
<td>19</td>
<td>22</td>
<td>11</td>
</tr>
<tr>
<td>20</td>
<td>23</td>
<td>8</td>
</tr>
</tbody>
</table>

The above data set contains the member incidence information or the joint connectivity data for each member. This completes the geometry of the structure.

START USER TABLE

This command is utilized to set up a user-provided steel table. All user-provided steel tables must start with this command.

TABLE 1

Each table needs an unique numerical identification. The above command starts setting up Table no. 1. Up to twenty tables may be specified per run.

UNIT CM
WIDE FLANGE

This command is used to specify the section-type as WIDE FLANGE in this table. Note that several section-types such as WIDE FLANGE, CHANNEL, ANGLE, TEE etc. are available for specification (See section 5.19 of the Technical Reference Manual (on page 2446)).

<table>
<thead>
<tr>
<th>Beam</th>
<th>Area</th>
<th>Depth</th>
<th>Moments of Inertia</th>
</tr>
</thead>
<tbody>
<tr>
<td>MED250</td>
<td>32.2</td>
<td>25.5</td>
<td>0.6 10.2 0.85 3400 150 6.2 15.7 15.4</td>
</tr>
<tr>
<td>MED300</td>
<td>47.5</td>
<td>30.4</td>
<td>0.72 12.4 1.07 7160 335 14.3 21.9 23.8</td>
</tr>
<tr>
<td>MED350</td>
<td>64.6</td>
<td>35.6</td>
<td>0.73 17.2 1.15 14150 970 22.8 26.0 35.5</td>
</tr>
</tbody>
</table>

The above data set is used to specify the properties of three wide flange sections. The data for each section consists of two parts. In the first line, the section-name is provided. You are allowed to provide any section name within twelve characters. The second line contains the section properties required for the particular section-type. Each section-type requires a certain number of data (area of cross-section, depth, moment of inertias etc.) provided in a certain order. For example, in this case, for wide flanges, ten different properties are required.

TABLE 2
UNIT CM
ANGLES
L30305 | 3.0 3.0 0.5 0.58 1.0 1.0 |
L40405 |
The above command and data lines set up another user provided table consisting of angle sections.

**END**

This command signifies the end of the user provided table data set. All user provided table related input must be terminated with this command.

**MEMBER PROPERTIES**

1 3 4 UPT 1 BEAM350
2 UPT 1 BEAM300 ; 5 6 7 UPT 1 BEAM250
8 TO 13 UPT 1 BEAM250
14 TO 19 UPT 2 L30305
20 TO 23 UPT 2 L40405

In the above command lines, the member properties are being assigned from the user provided tables created earlier. The word UPT signifies that the properties are from the user-provided table. This is followed by the table number and then the section name as specified in the user-provided table. The numbers 1 or 2 following the word UPT indicate the table from which section names are fetched.

**MEMBER TRUSS**

14 TO 23

The above command is used to designate members 14 to 23 as truss members.

**MEMBER RELEASE**

5 START MZ

The MEMBER RELEASE command is used to release the MZ moment at the start joint of member no. 5.

**UNIT MMS**

**DEFINE MATERIAL START**

ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4

The above command set is used to specify modulus of elasticity, density, Poisson’s ratio and beta angle values. Built-in default value of steel is used for the material constants.

**UNIT KNS METER**

The force unit is reset to KNS, and length unit to METER using this command.

**SUPPORT**

1 FIXED ; 2 PINNED

The above command set is used to designate supports. Here, joint 1 is designated as a fixed support and joint 2 is designated as a pinned support.

**LOADING 1 DEAD AND LIVE LOAD**

SELFWEIGHT Y -1.0
The above command set is used to specify the loadings on the structure. In this case, dead and live loads are provided through load case 1. It consists of selfweight, concentrated loads at joints 4, 5 and 11, and distributed loads on some members.

This command instructs the program to execute the analysis at this point.

The above commands are used to specify parameters for steel design.

This command will perform a code check on members 3 and 19 per the Eurocode 3 steel design code with the UK National Annex. A member selection too is performed for member 20. For each member, the member selection will be performed from the table that was originally used for the specification of the member property. In this case, the selection will be from the respective user tables from which the properties were initially assigned. It may be noted that properties may be provided (and selection may be performed) from built-in steel tables and user provided tables in the same data file.

This command terminates a STAAD run.

Input File

STAAD PLANE EXAMPLE FOR USER TABLE
UNIT METER KNS
JOINT COORD
1 0.0 ; 2 9 0 ; 3 0 6 0 6 9 6 0
7 0 10.5 ; 8 9 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
14 3.0 12.3 ; 15 6.0 12.3 ; 16 4.5 13.2
MEMBER INCIDENCES
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
START USER TABLE
TABLE 1
UNIT CM
WIDE FLANGE
BEAM250
32.2 25.5 0.6 10.2 0.85 3400 150 6.2 15.7 15.4
BEAM300
47.5 30.4 0.72 12.4 1.07 7160 335 14.3 21.9 23.8
Application Examples
EX. British Design Examples

BEAM350
64.6 35.6 0.73 17.2 1.15 14150 970 22.8 26.0 35.5

TABLE 2
UNIT CM
ANGLES
L30305
3.0 3.0 0.5 0.58 1.0 1.0
L40405
4.0 4.0 0.5 0.78 1.33 1.33
L50505
5.0 5.0 0.5 0.98 1.67 1.67

END

MEMBER PROPERTIES
1 3 4 UPT 1 BEAM350
2 UPT 1 BEAM300 ; 5 6 7 UPT 1 BEAM250
8 TO 13 UPT 1 BEAM250
14 TO 19 UPT 2 L30305
20 TO 23 UPT 2 L40405
*MEMBER TRUSS
*14 TO 23
MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
UNIT KNS METER
SUPPORT
1 FIXED ; 2 PINNED
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -65. ; 11 FY -155
MEMB LOAD
8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
PERFORM ANALYSIS
PARAMETER
CODE EN 1993-1-1:2005
NA 1
BEAM 3 ALL
NSF 0.85 ALL
KY 1.2 MEMB 3 4
CHECK CODE MEMBER 3 19
SELECT MEMB 20
FINISH
1. STAAD PLANE EXAMPLE FOR USER TABLE

INPUT FILE: UK-17 User-Provided Tables.STD

2. UNIT METER KNS

3. JOINT COORD
4. 1 0 0 ; 2 9 0 ; 3 0 6 0 6 9 6 0
5. 7 0 10.5 ; 8 9 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
6. 11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
7. 14 3.0 12.3 ; 15 6.0 12.3 ; 16 4.5 13.2

8. MEMBER INCIDENCES
9. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
10. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
11. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
12. 14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
13. 18 10 15 ; 19 10 13 ; 20 7 9
14. 21 9 11 ; 22 10 11 ; 23 8 10

15. START USER TABLE

16. TABLE 1

17. UNIT CM

18. WIDE FLANGE
19. BEAM250
20. 32.2 25.5 0.6 10.2 0.85 3400 150 6.2 15.7 15.4
21. BEAM300
22. 47.5 30.4 0.72 12.4 1.07 7160 335 14.3 21.9 23.8
23. BEAM350
24. 64.6 35.6 0.73 17.2 1.15 14150 970 22.8 26.0 35.5

25. TABLE 2

26. UNIT CM

27. ANGLES
28. L30305
29. 3.0 3.0 0.5 0.58 1.0 1.0
30. L40405
31. 4.0 4.0 0.5 0.78 1.33 1.33
32. L50505
33. 5.0 5.0 0.5 0.98 1.67 1.67

34. END

35. MEMBER PROPERTIES
36. 1 3 4 UPT 1 BEAM350
37. 2 UPT 1 BEAM300 ; 5 6 7 UPT 1 BEAM250
38. 8 TO 13 UPT 1 BEAM250

EXAMPLE FOR USER TABLE

- - PAGE NO. 2

39. 14 TO 19 UPT 2 L30305
40. 20 TO 23 UPT 2 L40405
41. *MEMBER TRUSS
42. *14 TO 23
43. MEMB RELEASE
44. 5 START MZ
45. 14 TO 23 START MPY 0.99 MPZ 0.99
46. 14 TO 23 END MPY 0.99 MPZ 0.99
47. UNIT MMS
48. DEFINE MATERIAL START
49. ISOTROPIC STEEL
50. E 210
51. POISSON 0.3
52. DENSITY 7.68E-008
53. ALPHA 6E-006
54. DAMP 0.03
55. TYPE STEEL
56. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
57. END DEFINE MATERIAL
58. CONSTANTS
59. MATERIAL STEEL ALL
60. BETA 90.0 MEMB 3 4
61. UNIT KNS METER
62. SUPPORT
63. 1 FIXED ; 2 PINNED
64. LOADING 1 DEAD AND LIVE LOAD
65. SELFWEIGHT Y -1.0
66. JOINT LOAD
67. 4 5 FY -65. ; 11 FY -155
68. MEMB LOAD
69. 8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
70. PERFORM ANALYSIS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS         16  NUMBER OF MEMBERS      23
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      43
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
71. PARAMETER
72. CODE EN 1993-1-1:2005
    EXAMPLE FOR USER TABLE -- PAGE NO.    3
73. NA 1
74. BEAM 3 ALL
75. NSF 0.85 ALL
76. KY 1.2 MEMB 3 4
77. CHECK CODE MEMBER 3 19
STEEL DESIGN

STAAD.PRO CODE CHECKING - BS EN 1993-1-1:2005
********************************************
NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
PROGRAM CODE REVISION V1.13 BS_EC3_2005/1
    EXAMPLE FOR USER TABLE -- PAGE NO.    4
ALL UNITS ARE - KNS METER (UNLESS OTHERWISE Noted)
MEMBER TABLE RESULT/ CRITICAL COND/ RATIO/ LOADING/
          FX  MY  MZ  LOCATION
=======================================================================
*     3 ST BEAM350   (UPT)
FAIL   EC-6.3.3-662  2.339
**EX. UK-18 Stress Calculation for Plate Elements**

This is an example which demonstrates the calculation of principal stresses on a finite element.

This problem is installed with the program by default to
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-18
Stress Calculation for Plate Elements.STD when you install the program.
Where:

$L = 3 \text{ m}$

Fixed Supports at Joints 1, 2, 3, 4, 5, 9, 13

Load intensity = 2000 Kn/sq.m in -Y direction

Actual input is shown in bold lettering followed by explanation.

STAAD SPACE SAMPLE CALCULATION FOR
* ELEMENT STRESSES

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D structure).

UNIT METER KNS

Defines the input units for the data that follows.

JOINT COORDINATES
1 0 0 4 9 0 0
REPEAT 3 0 0 3

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command is used to generate coordinates of joints 5 to 16 based on the pattern of joints 1 to 4.

ELEMENT INCIDENCE
1 1 5 6 2 TO 3
REPEAT 2 3 4
Element connectivities of elements 1 to 3 are defined first, based on which, the connectivities of elements 4 to 9 are generated.

```plaintext
ELEMENT PROPERTIES
1 TO 9 THICK 0.25
```

Elements 1 to 9 have a thickness of 0.25 m.

```plaintext
UNIT MMS
DEFINE MATERIAL ISOTROPIC CONCRETE E 21.0 POISSON 0.17 DENSITY 2.36158e-008 ALPHA 5e-006 DAMP 0.05 G 9.25 TYPE CONCRETE STRENGTH FCU 0.0275 END DEFINE MATERIAL CONSTANTS MATERIAL CONCRETE ALL UNIT METER
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```plaintext
SUPPORT 1 TO 4 5 9 13 FIXED
```

Fixed support conditions exist at the above mentioned joints.

```plaintext
LOAD 1 ELEMENT LOAD 1 TO 9 PRESSURE -2000.0
```

A uniform pressure of 2000 Kn/sq.m is applied on all the elements. In the absence of an explicit direction specification, the load is assumed to act along the local Z axis. The negative value indicates that the load acts opposite to the positive direction of the local Z.

```plaintext
PERFORM ANALYSIS
```

The above command instructs the program to proceed with the analysis.

```plaintext
PRINT SUPPORT REACTION
```

The above command is self-explanatory.

```plaintext
UNIT MMS PRINT ELEMENT STRESSES LIST 4
```

Element stresses at the centroid of the element are printed using the above command. The output includes membrane stresses, shear stresses, bending moments per unit width and principal stresses. The change of length unit from meter to mms indicates that the values will be printed in KN and MMs units.

```plaintext
FINISH
```

The STAAD run is terminated.

**Calculation of principal stresses for element 4**

Calculations are presented for the top surface only.
\[ SX = 0.0 \text{kN/mm}^2 \]
\[ SY = 0.0 \text{kN/mm}^2 \]
\[ SXY = 0.0 \text{kN/mm}^2 \]
\[ MX = 2,111.84 \text{kN-mm/mm} \]
\[ MY = 10,726.22 \text{kN-mm/m} \]
\[ MXY = 4,553.97 \text{kN-mm/mm} \]
\[ S = \frac{1}{6t^2} = \frac{1}{(6 \cdot 250^2)} = 10,416.67 \text{mm}^2 \] (Section Modulus)
\[ \sigma_x = SX + \frac{MX}{S} = 0.0 + \frac{2,111.84}{10,416.67} = 0.2027 \text{kN/mm} \]
\[ \sigma_y = SY + \frac{MY}{S} = 0.0 + \frac{10,726.22}{10,416.67} = 1.0297 \text{kN/mm} \]
\[ \tau_{xy} = SXY + \frac{MXY}{S} = 0.0 + \frac{4,553.97}{10,416.67} = 0.4372 \text{kN/mm} \]
\[ T_{Max} = \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2} = \sqrt{\frac{(0.2027 - 1.0297)^2}{4} + 0.4372^2} = 0.6018 \text{KN/mm}^2 \]
\[ S_{Max} = \frac{(\sigma_x + \sigma_y)}{2} + T_{Max} = (0.2027 + 1.0297)/2 + 0.6018 = 1.218 \text{kN/mm}^2 \]
Say 1.22 kN/mm²
\[ S_{Min} = \frac{(\sigma_x + \sigma_y)}{2} - T_{Max} = (0.2027 + 1.0297)/2 - 0.6018 = 0.0144 \text{kN/mm}^2 \]
Say 0.01 kN/mm²

\[ \text{Angle} = \frac{1}{2} \tan^{-1}\left(\frac{2\tau_{xy}}{\sigma_x - \sigma_y}\right) = \frac{1}{2} \tan^{-1}\left(\frac{2(0.4372)}{0.2027 - 1.0297}\right) = -23.30^\circ \]
\[ \text{VON} = 0.707\sqrt{(S_{Max} - S_{Min})^2 + S_{Max}^2 + S_{Min}^2} = 0.707\sqrt{(1.218 - 0.0144)^2 + 1.218^2 + 0.0144^2} = 1.2107 \text{kN/mm}^2 \]

**Input File**

STAAD SPACE SAMPLE CALCULATION FOR
* ELEMENT STRESSES
UNIT METER KNS
JOINT COORDINATES
1 0 0 0 4 9 0 0
REPEAT 3 0 0 3
ELEMENT INCIDENCE
1 1 5 6 2 TO 3
REPEAT 2 3 4
ELEMENT PROPERTIES
1 TO 9 THICK 0.25
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
SUPPORT
1 TO 4 5 9 13 FIXED
LOAD 1
ELEMENT LOAD
1 TO 9 PRESSURE -2000.0
PERFORM ANALYSIS
PRINT SUPPORT REACTION
UNIT MMS
PRINT ELEMENT STRESSES LIST 4
FINISH

STAAD Output File

****************************************************
**                                                 *
**           STAAD.Pro CONNECT Edition             *
**           Version  22.01.00.**                   *
**           Proprietary Program of                *
**           Bentley Systems, Inc.                  *
**           Date=    APR 14, 2019                  *
**           Time=    22:53:23                      *
**                                                 *
**  Licensed to: Bentley Systems Inc                *
****************************************************

1. STAAD SPACE SAMPLE CALCULATION FOR
INPUT FILE: UK-18 Stress Calculation for Plate Elements.STD
2. * ELEMENT STRESSES
3. UNIT METER KNS
4. JOINT COORDINATES
5. 1 0 0 0 4 9 0 0
6. REPEAT 3 0 0 3
7. ELEMENT INCIDENCE
8. 1 1 5 6 2 TO 3
9. REPEAT 2 3 4
10. ELEMENT PROPERTIES
11. 1 TO 9 THICK 0.25
12. UNIT MMS
13. DEFINE MATERIAL START
14. ISOTROPIC CONCRETE
15. E 21.0
16. POISSON 0.17
17. DENSITY 2.36158E-008
18. ALPHA 5E-006
19. DAMP 0.05
20. G 9.25
21. TYPE CONCRETE
22. STRENGTH FCU 0.0275
23. END DEFINE MATERIAL
24. CONSTANTS
25. MATERIAL CONCRETE ALL
26. UNIT MMS
27. SUPPORT
28. 1 TO 4 5 9 13 FIXED
29. LOAD 1
30. ELEMENT LOAD
31. 1 TO 9 PRESSURE -2000.0
32. PERFORM ANALYSIS
   SAMPLE CALCULATION FOR -- PAGE NO.  2
* ELEMENT STRESSES

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS     16  NUMBER OF MEMBERS     0
NUMBER OF PLATES      9   NUMBER OF SOLIDS      0
NUMBER OF SURFACES   0    NUMBER OF SUPPORTS   7
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      54
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
33. PRINT SUPPORT REACTION

SUPPORT REACTION
   SAMPLE CALCULATION FOR -- PAGE NO.  3
* ELEMENT STRESSES
SUPPORT REACTIONS -UNIT KNS  METE  STRUCTURE TYPE = SPACE
-----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM-Z
1    1      0.00   -1291.48      0.00   -348.06      0.00    348.06
2    1      0.00    8700.82      0.00  -26485.08      0.00   -384.11
3    1      0.00    3742.16      0.00  -87763.47      0.00   3265.14
4    1      0.00    3502.76      0.00 -66094.57      0.00  -23952.47
5    1      0.00    8700.82      0.00    384.11      0.00  26485.08
9    1      0.00    3742.16      0.00  -3265.14      0.00  87763.47
13   1      0.00    3502.76      0.00  23952.47      0.00  66094.57
************** END OF LATEST ANALYSIS RESULT **************
34. UNIT MMS
35. PRINT ELEMENT STRESSES LIST 4

ELEMENT STRESSES LIST     4
   SAMPLE CALCULATION FOR -- PAGE NO.  4
* ELEMENT STRESSES
ELEMENT STRESSES    FORCE,LENGTH UNITS= KNS  MMS
----------------
STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
ELEMENT LOAD    SQX       SQY       MX       MY       MXY
                 VONT      VONB      SX       SY       SXY
                 TRESCAT    TRESCAB
4    1         0.02       -0.02     2080.05    10634.76     4648.47
      1.21       1.21          0.00          0.00          0.00
      1.22       1.22
TOP : SMAX=     1.22 SMIN=     0.00 TMAX=     0.61 ANGLE= 66.3
BOTT: SMAX=    -0.00 SMIN=   -1.22 TMAX=     0.61 ANGLE=-23.7
**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
MAXIMUM
PRINCIPAL
STRESS
PRINCIPAL
SHAP
VONMISES
TRESCA
1.216739E+00 -1.216739E+00  6.064286E-01  1.214803E+00  1.216739E+00
PLATE NO.   4     4     4     4     4
CASE NO.    1     1     1     1     1
***************END OF ELEMENT FORCES***************
36. FINISH
   SAMPLE CALCULATION FOR -- PAGE NO.  5
* ELEMENT STRESSES
********** END OF THE STAAD.Pro RUN **********
EX. UK-19 Inclined Supports

This example demonstrates the usage of inclined supports. The word INCLINED refers to the fact that the restraints at a joint where such a support is specified are along a user-specified axis system instead of along the default directions of the global axis system. STAAD.Pro offers a few different methods for assigning inclined supports, and we examine those in this example.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-19 Inclined Supports.STD when you install the program.

![Figure 502: Example Problem No. 19](image)

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE
INPUT WIDTH 79
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y, and Z coordinates.

```
UNIT METER KN
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;
```

Joint number followed by X, Y and Z coordinates are provided above.

**Note:** Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.
Defines the members by the joints to which they are connected.

```
UNIT MMS KN
MEMBER PROPERTY AMERICAN
 4 5 PRIS YD 800
 1 TO 3 PRIS YD 750 ZD 500
```

Properties for all members of the model are provided using the PRISMATIC option (abbreviated to PRIS here). YD and ZD stand for depth and width. If ZD is not provided, a circular shape with diameter = YD is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. The values are provided in MMS unit.

```
DEFINE MATERIAL START
  ISOTROPIC CONCRETE
  E 21.72
  POISSON 0.17
  DENSITY 2.35615e-008
  ALPHA 5e-006
  DAMP 0.05
  G 9.281
  TYPE CONCRETE
  STRENGTH FCU 0.0275
END DEFINE MATERIAL
```

Material constants like E (modulus of elasticity) and Poisson's ratio are specified following the command CONSTANTS.

```
UNIT METER KN
SUPPORTS
  5 INCLINED REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
  6 INCLINED REFJT 3 FIXED BUT MX MY MZ KFX 30000
  1 PINNED
  4 INCLINED 1 0 1 FIXED BUT FX MX MY MZ
```

We assign supports (restraints) at 4 nodes - 5, 6, 1 and 4. For 3 of those, namely, 5, 6 and 4, the node number is followed by the keyword INCLINED, signifying that an INCLINED support is defined there. For the remaining one - node 1 - that keyword is missing. Hence, the support at node 1 is a global direction support.

The most important aspect of inclined supports is their axis system. Each node where an inclined support is defined has its own distinct local X, local Y and local Z axes. In order to define the axis system, we first have to define a datum point. The support node and the datum point together help define the axis system.

Three different methods are shown in the above 3 instances for defining the datum point.

- At node 5, notice the keyword REF followed by the numbers (10,5,10). This means that the datum point associated with node 5 is one which has the global coordinates of (10m, 5m, 10m). Coincidentally, this happens to be node 2.
- At node 6, the keyword REFJT is used followed by the number 3. This means that the datum point for support node 6 is the joint number 3 of the model. The coordinates of the datum point are hence those of node 3, namely, (20m, 5m and 20m).
- At node 4, the word INCLINED is merely followed by 3 numbers (1,0,1). In the absence of the words REF and REFJT, the program sets the datum point to be the following. It takes the coordinates of node 4, which are (30m,5m,30m) and adds to them, the 3 numbers which comes after the word INCLINED. Thus, the datum point becomes (31m, 5m and 31m).
Once the datum point is established, the local axis system is defined as follows. Local X is a straight line (vector) pointing from the support node towards the datum point. Local Z is the vector obtained by the cross product of local X and the global Y axis (unless the SET Z UP command is used in which case one would use global Z instead of global Y and that would yield local Y). Local Y is the vector resulting from the cross product of local Z and local X. The right hand rule must be used when performing these cross products.

Notice the unique nature of these datum points. The one for node 5 tells us that a line connecting nodes 5 to 2 is the local X axis, and is hence along the axis of member 4. By defining a KFX spring at that one, we are saying that the lower end of member 4 can move along its axis like the piston of a car engine. Think of a pile bored into rock with a certain amount of freedom to expand and contract axially.

The same is true for the support at the bottom of member 5. The local X axis of that support is along the axis of member 5. That also happens to be the case for the supported end of member 3. The line going from node 4 to the datum point (31,5,31) happens to be coincident with the axis of the member, or the traffic direction. The expression FIXED BUT FX MX MY MZ for that support indicates that it is free to translate along local X, suggesting that it is an expansion joint - free to expand or contract along the axis of member 3.

Since MX, MY, and MZ are all released at these supports, no moment will be resisted by these supports.

LOAD 1 DEAD LOAD
SELFWEIGHT Y -1.2
LOAD 2 LIVE LOAD
MEMBER LOAD
1 TO 3 UNI GY -6
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK

Three load cases followed by the instruction for the type of analysis are specified. The PRINT STATICS CHECK option will instruct the program to produce a report consisting of total applied load versus total reactions from the supports for each primary load case.

PRINT SUPPORT REACTION

By default, support reactions are printed in the global axis directions. The above command is an instruction for such a report.

SET INCLINED REACTION
PRINT SUPPORT REACTION

Just earlier, we saw how to obtain support reactions in the global axis system. What if we need them in the inclined axis system? The "SET INCLINED REACTION" is a switch for that purpose. It tells the program that reactions should be reported in the inclined axis system instead of the global axis system. This has to be followed by the PRINT SUPPORT REACTIONS command.

PRINT MEMBER FORCES
PRINT JOINT DISP
FINISH

Member forces are reported in the local axis system of the members. Joint displacements at all joints are reported in the global axis system. Following this, the STAAD run is terminated.

**Input File**

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;
```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3;
UNIT MMS KN
MEMBER PROPERTY AMERICAN
4 5 PRIS YD 800
1 TO 3 PRIS YD 750 ZD 500
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER KN
SUPPORTS
5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
1 PINNED
4 INC 1 0 1 FIXED BUT FX MX MY MZ
LOAD 1 DEAD LOAD
SELFWEIGHT Y -1.2
LOAD 2 LIVE LOAD
MEMBER LOAD
1 TO 3 UNI GY -6
LOAD COMB 3
1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PRINT SUPPORT REACTION
SET INCLINED REACTION
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
PRINT JOINT DISP
FINISH

STAAD Output File

******************************************************************************************
*
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 22:53:28 *
*
* Licensed to: Bentley Systems Inc *
******************************************************************************************

1. STAAD SPACE
INPUT FILE: UK-19 Inclined Supports.STD
2. INPUT WIDTH 79
3. UNIT METER KN
4. JOINT COORDINATES
   1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25

5. MEMBER INCIDENCES
   1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3

6. UNIT MMS KN

7. MEMBER PROPERTY AMERICAN
   4 5 PRIS YD 800
   1 TO 3 PRIS YD 750 ZD 500

8. DEFINE MATERIAL START
   ISOTROPIC CONCRETE
   E 21.0
   POISSON 0.17
   DENSITY 2.36158E-008
   ALPHA 5E-006
   DAMP 0.05
   G 9.25
   TYPE CONCRETE
   STRENGTH FCU 0.0275
   END DEFINE MATERIAL

9. CONSTANTS
   MATERIAL CONCRETE ALL
   UNIT METER KN

10. SUPPORTS
    5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
    6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
    1 PINNED
    4 INC 1 0 1 FIXED BUT FX MX MY MZ
    LOAD 1 DEAD LOAD
    SELFWEIGHT Y -1.2
    LOAD 2 LIVE LOAD
    MEMBER LOAD
    1 TO 3 UNI GY -6
    LOAD COMB 3
    1 1.0 2 1.0

11. LOAD 1 DEAD LOAD
12. SELFWEIGHT Y -1.2
13. LOAD 2 LIVE LOAD
14. MEMBER LOAD
15. 1 TO 3 UNI GY -6
16. LOAD COMB 3
17. 1 1.0 2 1.0

18. PERFORM ANALYSIS PRINT STATICS CHECK

Problem Statistics
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      27
TOTAL LOAD COMBINATION CASES =    1

Static Load/Reaction/Equilibrium Summary for Case No. 1
DEAD LOAD
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
   X =  0.150000000E+02
   Y =  0.411580006E+01
   Z =  0.150000000E+02

Total Applied Load 1
***Total Applied Load ( KN   METE ) Summary (Loading  1 )
   SUMMATION FORCE-X =     0.00
   SUMMATION FORCE-Y =     -697.60
   SUMMATION FORCE-Z =     0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 10463.94  MY= 0.00  MZ= -10463.94
TOTAL REACTION LOAD 1
***TOTAL REACTION LOAD( KN METE ) SUMMARY (LOADING 1)
SUMMATION FORCE-X = -0.00
SUMMATION FORCE-Y = 697.60
SUMMATION FORCE-Z = -0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -10463.94  MY= -0.00  MZ= 10463.94
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING 1)
MAXIMUMS AT NODE
X = -8.02030E-01  5
Y = -2.50768E+00  3
Z = -8.02030E-01  5
RX= 8.02194E-18    1
RZ= 2.71938E-03    4

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 2
LIVE LOAD
CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.149999999E+02
Y = 0.500000017E+01
Z = 0.149999999E+02

TOTAL APPLIED LOAD 2
***TOTAL APPLIED LOAD ( KN METE ) SUMMARY (LOADING 2)
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -254.56
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 3818.38  MY= 0.00  MZ= -3818.38
TOTAL REACTION LOAD 2
***TOTAL REACTION LOAD( KN METE ) SUMMARY (LOADING 2)
SUMMATION FORCE-X = -0.00
SUMMATION FORCE-Y = 254.56
SUMMATION FORCE-Z = -0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -3818.38  MY= -0.00  MZ= 3818.38
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING 2)
MAXIMUMS AT NODE
X = -2.97766E-01  5
Y = -9.34280E-01  3
Z = -2.97766E-01  5
RX= -3.94177E-18  4
RZ= 2.14818E-03   4

************* END OF DATA FROM INTERNAL STORAGE *************
39. PRINT SUPPORT REACTION
SUPPORT REACTION
STAAD SPACE
SUPPORT REACTIONS -UNIT KN METE STRUCTURE TYPE = SPACE
----------
JOINT LOAD FORCE-X FORCE-Y FORCE-Z MOM-X MOM-Y MOM Z
5 1 216.27 289.28 216.27 0.00 0.00 0.00
2 86.61 94.78 86.61 0.00 0.00 0.00
3 302.88 384.06 302.88 0.00 0.00 0.00
6 1 -213.07 287.50 -213.07 0.00 0.00 0.00
### Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>416.75</td>
<td>94.06</td>
<td>-85.33</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-298.40</td>
<td>381.56</td>
<td>-298.40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1</td>
<td>-3.20</td>
<td>60.34</td>
<td>-3.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-1.28</td>
<td>32.84</td>
<td>-1.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-4.48</td>
<td>93.18</td>
<td>-4.48</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>60.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**END OF LATEST ANALYSIS RESULT**

40. SET INCLINED REACTION

41. PRINT SUPPORT REACTION

**SUPPORT REACTION**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>416.75</td>
<td>60.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>152.83</td>
<td>67.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>564.85</td>
<td>67.90</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1</td>
<td>-3.20</td>
<td>60.34</td>
<td>-3.20</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-1.28</td>
<td>32.84</td>
<td>-1.28</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-4.48</td>
<td>93.18</td>
<td>-4.48</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>60.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**END OF LATEST ANALYSIS RESULT**

42. PRINT MEMBER FORCES

**MEMBER FORCES**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-4.52</td>
<td>60.34</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>2</td>
<td>4.52</td>
<td>89.95</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-209.31</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.81</td>
<td>52.02</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-135.62</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-6.33</td>
<td>93.18</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-344.93</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>6.33</td>
<td>141.96</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-73.16</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>118.73</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>207.05</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>116.41</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-190.59</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>0.00</td>
<td>89.82</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>207.54</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>60.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>51.97</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>134.91</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>141.79</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>342.45</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>93.36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>416.75</td>
<td>59.61</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>2</td>
<td>-345.52</td>
<td>41.11</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>80.12</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-154.72</td>
<td>-6.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-154.72</td>
<td>-6.67</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>JOINT</td>
<td>LOAD</td>
<td>X-TRANS</td>
<td>Y-TRANS</td>
<td>Z-TRANS</td>
<td>X-ROTAN</td>
<td>Y-ROTAN</td>
<td>Z-ROTAN</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0027</td>
<td>0.0000</td>
<td>-0.0027</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0012</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0039</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0039</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-0.0377</td>
<td>-2.5077</td>
<td>-0.0377</td>
<td>0.0007</td>
<td>0.0000</td>
<td>0.0007</td>
</tr>
<tr>
<td>2</td>
<td>-0.0151</td>
<td>-0.9157</td>
<td>-0.0151</td>
<td>0.0003</td>
<td>0.0000</td>
<td>0.0003</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.0008</td>
<td>-3.3771</td>
<td>0.0008</td>
<td>0.0011</td>
<td>0.0000</td>
<td>0.0011</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-0.0377</td>
<td>0.0000</td>
<td>-0.0377</td>
<td>-0.0027</td>
<td>0.0000</td>
<td>-0.0027</td>
</tr>
<tr>
<td>2</td>
<td>-0.0151</td>
<td>0.0000</td>
<td>-0.0151</td>
<td>-0.0012</td>
<td>0.0000</td>
<td>-0.0012</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.0528</td>
<td>0.0000</td>
<td>-0.0528</td>
<td>-0.0039</td>
<td>0.0000</td>
<td>-0.0039</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-0.8020</td>
<td>-0.8020</td>
<td>-0.8020</td>
<td>0.0024</td>
<td>0.0000</td>
<td>-0.0024</td>
</tr>
<tr>
<td>2</td>
<td>-0.2978</td>
<td>-0.2978</td>
<td>-0.2978</td>
<td>0.0007</td>
<td>0.0000</td>
<td>-0.0007</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-1.0998</td>
<td>-1.0998</td>
<td>-1.0998</td>
<td>0.0031</td>
<td>0.0000</td>
<td>-0.0031</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.7929</td>
<td>0.7929</td>
<td>0.7929</td>
<td>-0.0024</td>
<td>0.0000</td>
<td>-0.0024</td>
</tr>
<tr>
<td>2</td>
<td>0.2941</td>
<td>-0.2941</td>
<td>0.2941</td>
<td>-0.0008</td>
<td>0.0000</td>
<td>-0.0008</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1.0871</td>
<td>-1.0871</td>
<td>1.0871</td>
<td>0.0032</td>
<td>0.0000</td>
<td>0.0032</td>
<td></td>
</tr>
</tbody>
</table>

Related Links

- **M. To assign an inclined support** (on page 817)
- **TR.27.2 Inclined Support Specification** (on page 2516)
- **Create Support dialog** (on page 2983)
EX. UK-20 Generating a Structure in Cylindrical Coordinates

This example generates the geometry of a cylindrical tank structure using the cylindrical coordinate system. The tank lies on its side in this example.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-20 Generating a Structure in Cylindrical Coordinates.STD when you install the program.

In this example, a cylindrical tank is modeled using finite elements. The radial direction is in the XY plane and longitudinal direction is along the Z-axis. Hence, the coordinates in the XY plane are generated using the cylindrical coordinate system.

STAAD SPACE

UNIT METER KN

The type of structure (space frame) and length and force units for data to follow are specified.

JOINT COORD CYLINDRICAL

The above command instructs the program that the coordinate data that follows is in the cylindrical coordinate system (r,theta,z).

1 3.5 0 0 8 3.5 315 0

Joint 1 has an 'r' of 3.5 meters, theta of 0 degrees and Z of 0 ft. Joint 8 has an 'r' of 3.5 meters, theta of 315 degrees and Z of 0 ft. The 315 degrees angle is measured counter-clockwise from the +ve direction of the X-axis. Joints 2 to 7 are generated by equal increments of the coordinate values between joints 1 and 8.

REPEAT 2 0 0 3.0
The REPEAT command is used to generate joints 9 through 24 by repeating twice, the pattern of joints 1 to 8 at Z-increments of 3.0 meters for each REPEAT.

<table>
<thead>
<tr>
<th>PRINT JOINT COORD</th>
</tr>
</thead>
</table>

The above command is used to produce a report consisting of the coordinates of all the joints in the Cartesian coordinate system. Note that even though the input data was in the cylindrical coordinate system, the output is in the Cartesian coordinate system.

<table>
<thead>
<tr>
<th>ELEMENT INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 10 9 TO 7 1 1</td>
</tr>
<tr>
<td>8 8 1 9 16</td>
</tr>
<tr>
<td>REPEAT ALL 1 8 8</td>
</tr>
</tbody>
</table>

The above 4 lines identify the element incidences of all 16 elements. Incidences of element 1 is defined as 1 2 10 9. Incidences of element 2 is generated by incrementing the joint numbers of element 1 by 1, incidences of element 3 is generated by incrementing the incidences of element 2 by 1 and so on up to element 7. Incidences of element 8 has been defined above as 8 1 9 16. The REPEAT ALL command states that the pattern of all the elements defined by the previous 2 lines, namely elements 1 to 8, must be repeated once with an element number increment of 8 and a joint number increment of 8 to generate elements 9 through 16.

<table>
<thead>
<tr>
<th>PRINT ELEMENT INFO</th>
</tr>
</thead>
</table>

The above command is self-explanatory.

<table>
<thead>
<tr>
<th>FINISH</th>
</tr>
</thead>
</table>

**Input File**

<table>
<thead>
<tr>
<th>STAAD SPACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNIT METER KN</td>
</tr>
<tr>
<td>JOINT COORD CYLINDRICAL</td>
</tr>
<tr>
<td>1 3.5 0 0 8 3.5 315 0</td>
</tr>
<tr>
<td>REPEAT 2 0 0 3.0</td>
</tr>
<tr>
<td>PRINT JOINT COORD</td>
</tr>
<tr>
<td>ELEMENT INCIDENCES</td>
</tr>
<tr>
<td>1 1 2 10 9 TO 7 1 1</td>
</tr>
<tr>
<td>8 8 1 9 16</td>
</tr>
<tr>
<td>REPEAT ALL 1 8 8</td>
</tr>
<tr>
<td>PRINT ELEMENT INFO</td>
</tr>
<tr>
<td>FINISH</td>
</tr>
</tbody>
</table>

**STAAD Output File**

<table>
<thead>
<tr>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
</tr>
</tbody>
</table>

************************************************************************
* *
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.** *
* Proprietary Program of *
* Bentley Systems, Inc. *
* Date= APR 14, 2019 *
* Time= 22:53:38 *
* *
* Licensed to: Bentley Systems Inc *
************************************************************************

1. STAAD SPACE
INPUT FILE: UK-20 Generating a Structure in Cylindrical Coordinates.STD
2. UNIT METER KN
### Joint Coordinates

COORDINATES ARE METE UNIT

<table>
<thead>
<tr>
<th>JOINT</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.500</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>2.475</td>
<td>2.475</td>
<td>0.000</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>3.500</td>
<td>0.000</td>
</tr>
<tr>
<td>4</td>
<td>-2.475</td>
<td>2.475</td>
<td>0.000</td>
</tr>
<tr>
<td>5</td>
<td>-3.500</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>6</td>
<td>-2.475</td>
<td>-2.475</td>
<td>0.000</td>
</tr>
<tr>
<td>7</td>
<td>-0.000</td>
<td>-3.500</td>
<td>0.000</td>
</tr>
<tr>
<td>8</td>
<td>2.475</td>
<td>-2.475</td>
<td>0.000</td>
</tr>
<tr>
<td>9</td>
<td>3.500</td>
<td>0.000</td>
<td>3.000</td>
</tr>
<tr>
<td>10</td>
<td>2.475</td>
<td>2.475</td>
<td>3.000</td>
</tr>
<tr>
<td>11</td>
<td>0.000</td>
<td>3.500</td>
<td>3.000</td>
</tr>
<tr>
<td>12</td>
<td>-2.475</td>
<td>2.475</td>
<td>3.000</td>
</tr>
<tr>
<td>13</td>
<td>-3.500</td>
<td>0.000</td>
<td>3.000</td>
</tr>
<tr>
<td>14</td>
<td>-2.475</td>
<td>-2.475</td>
<td>3.000</td>
</tr>
<tr>
<td>15</td>
<td>-0.000</td>
<td>-3.500</td>
<td>3.000</td>
</tr>
<tr>
<td>16</td>
<td>2.475</td>
<td>-2.475</td>
<td>3.000</td>
</tr>
<tr>
<td>17</td>
<td>3.500</td>
<td>0.000</td>
<td>6.000</td>
</tr>
<tr>
<td>18</td>
<td>2.475</td>
<td>2.475</td>
<td>6.000</td>
</tr>
<tr>
<td>19</td>
<td>0.000</td>
<td>3.500</td>
<td>6.000</td>
</tr>
<tr>
<td>20</td>
<td>-2.475</td>
<td>2.475</td>
<td>6.000</td>
</tr>
<tr>
<td>21</td>
<td>-3.500</td>
<td>0.000</td>
<td>6.000</td>
</tr>
<tr>
<td>22</td>
<td>-2.475</td>
<td>-2.475</td>
<td>6.000</td>
</tr>
<tr>
<td>23</td>
<td>-0.000</td>
<td>-3.500</td>
<td>6.000</td>
</tr>
<tr>
<td>24</td>
<td>2.475</td>
<td>-2.475</td>
<td>6.000</td>
</tr>
</tbody>
</table>

******* END OF DATA FROM INTERNAL STORAGE *******

### Element Incidences

<table>
<thead>
<tr>
<th>INCIDENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 2 10 9 TO 7 1 1</td>
</tr>
<tr>
<td>8 8 1 9 16</td>
</tr>
</tbody>
</table>

### Element Information

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>INCIDENCES</th>
<th>THICK (METE)</th>
<th>POISS</th>
<th>E1/E2</th>
<th>G1/G2</th>
<th>AREA</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1 2 10</td>
<td>9</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>2</td>
<td>2 3 11</td>
<td>10</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>3</td>
<td>3 4 12</td>
<td>11</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>4</td>
<td>4 5 13</td>
<td>12</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>
EX. UK-21 Analysis of a Structure with Tension-Only Members

This example illustrates the modeling of tension-only members using the MEMBER TENSION command.
This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-21 Analysis of a Structure with Tension-Only Members.STD when you install the program.

It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of “active” members (and hence the stiffness matrix) is load case dependent.

![Figure 504: Example Problem No. 21](image)

where:

\[ L = 5.25 \text{ m}, \; H = 3.5 \text{ m} \]

Load case 1: \( P_1 = 45 \text{ kN} \) & \( P_2 = 70 \text{ kN} \)
Load case 2: \( P_3 = -45 \text{ kN} \) & \( P_4 = -70 \text{ kN} \)

**STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS**

The input data is initiated with the word STAAD. This structure is a PLANE frame.

**UNIT METER KNS**

Units for the commands to follow are defined above.

**JOINT COORDINATES**

1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
Joint coordinates of joints 1 to 6 are defined above.

```
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

Incidences of members 1 to 10 are defined.

```
MEMBER TENSION
6 TO 9
```

Members 6 to 9 are defined as tension-only members using the MEMBER TENSION command. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a compressive force, it is disabled from the structure and the analysis is carried out again with the modified structure.

```
MEMBER PROPERTY BRITISH
1 TO 10 TA ST UC152X152X30
```

All members have been assigned a UC section from the British table.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
MATERIAL STEEL ALL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. The length units have been changed from meters to millimeters to facilitate the input of these values.

```
SUPPORT
1 6 PINNED
```

The supports are defined above.

```
LOAD 1
JOINT LOAD
2 FX 70
3 FX 45
```

Load 1 is defined above and consists of joint loads at joints 2 and 3.

```
PERFORM ANALYSIS
CHANGE
MEMBER TENSION
6 TO 9
```

One or more among the members 6 to 9 may have been inactivated in the previous analysis.

```
LOAD 2
JOINT LOAD
4 FX -45
5 FX -70
```
Load case 2 is described above.

```
LOAD 3
REPEAT LOAD
  1 1.0 2 1.0
```

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously.

```
PRINT ANALYSIS RESULTS
FINI
```

The analysis results are printed and the run terminated.

**Input File**

```
STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
UNIT METER KNS
SET NL 3
JOINT COORDINATES
  1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
MEMBER INCIDENCES
  1 1 2 5
  6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
MEMBER TENSION
  6 TO 9
MEMBER PROPERTY BRITISH
  1 TO 10 TABLE SC152X152X30
UNIT MMS
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 210
  POISSON 0.3
  DENSITY 7.6977e-008
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORT
  1 6 PINNED
LOAD 1
  JOINT LOAD
  2 FX 70
  3 FX 45
PERFORM ANALYSIS
CHANGE
MEMBER TENSION
  6 TO 9
LOAD 2
  JOINT LOAD
  4 FX -45
  5 FX -70
PERFORM ANALYSIS
CHANGE
LOAD 3
```
1. STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS

INPUT FILE: UK-21 Analysis of a Structure with Tension-Only Members.STD

2. UNIT METER KNS

3. SET NL 3

4. JOINT COORDINATES
5. 1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0

6. MEMBER INCIDENCES
7. 1 1 2 5
8. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5

9. MEMBER TENSION

10. 6 TO 9

11. MEMBER PROPERTY BRITISH

12. 1 TO 10 TABLE ST UC152X152X30

13. UNIT MMS

14. DEFINE MATERIAL START
15. ISOTROPIC STEEL
16. E 210
17. POISSON 0.3
18. DENSITY 7.6977E-008
19. ALPHA 6E-006
20. DAMP 0.03
21. TYPE STEEL
22. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2

23. END DEFINE MATERIAL

24. CONSTANTS
25. MATERIAL STEEL ALL

26. SUPPORT
27. 1 6 PINNED
28. LOAD 1

29. JOINT LOAD
30. 2 FX 70
31. 3 FX 45

32. PERFORM ANALYSIS

EXAMPLE FOR TENSION-ONLY MEMBERS
NUMBER OF JOINTS          6  NUMBER OF MEMBERS      10
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
*** LOAD CASE  1 -- START ITERATION NO.   2
**NOTE-Tension/Compression converged after  2 iterations, Case=    1
33. CHANGE
34. MEMBER TENSION
35. 6 TO 9
36. LOAD 2
37. JOINT LOAD
38. 4 FX -45
39. 5 FX -70
40. PERFORM ANALYSIS
*** LOAD CASE  2 -- START ITERATION NO.   2
**NOTE-Tension/Compression converged after  2 iterations, Case=    2
41. CHANGE
42. LOAD 3
43. REPEAT LOAD
44. 1 1.0 2 1.0
45. PERFORM ANALYSIS
*** LOAD CASE  3 -- START ITERATION NO.   2
**NOTE-Tension/Compression converged after  2 iterations, Case=    3
46. CHANGE
47. LOAD LIST 1 2 3
EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    3
48. PRINT ANALYSIS RESULTS
ANALYSIS RESULTS
EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    4
JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
1    1     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0008
2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0005
3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0001
2    1     0.2355    0.0132    0.0000    0.0000    0.0000   -0.0004
2    -0.1609   -0.0464    0.0000    0.0000    0.0000    0.0004
3     0.0228    0.0000    0.0000    0.0000    0.0000   -0.0000
3    1     0.3633    0.0133    0.0000    0.0000    0.0000   -0.0002
2    -0.3341   -0.0594    0.0000    0.0000    0.0000    0.0004
3     0.0147    0.0000    0.0000    0.0000    0.0000    0.0000
4    1     0.3341   -0.0594    0.0000    0.0000    0.0000   -0.0004
2    -0.3633    0.0133    0.0000    0.0000    0.0000    0.0002
3    -0.0147    0.0000    0.0000    0.0000    0.0000    0.0000
5    1     0.1609   -0.0464    0.0000    0.0000    0.0000   -0.0004
2    -0.2355    0.0132    0.0000    0.0000    0.0000    0.0004
3    -0.0228    0.0000    0.0000    0.0000    0.0000    0.0000
6    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0008
3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0001
EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    5
SUPPORT REACTIONS -UNIT KNS  MMS     STRUCTURE TYPE = PLANE
-----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z   MOM-X   MOM-Y   MOM Z
1    1   -114.92   -106.67    0.00    0.00    0.00    0.00

STAAD.Pro  4811  User Manual
### EXAMPLE FOR TENSION-ONLY MEMBERS

MEMBER END FORCES  
STRUCTURE TYPE = PLANE

---

**All Units Are -- KNS  MMS  (Local)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-30.23</td>
<td>0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-30.23</td>
<td>-0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>106.67</td>
<td>-0.08</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.00</td>
<td>-0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>158.18</td>
<td>158.18</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.25</td>
<td>0.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>200.03</td>
<td>200.03</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>-0.00</td>
<td>-0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-137.99</td>
<td>-137.99</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-43.08</td>
<td>-43.08</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>0.25</td>
<td>0.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>545.92</td>
<td>545.92</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>44.79</td>
<td>-0.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>43.08</td>
<td>43.08</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>-44.79</td>
<td>0.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>545.92</td>
<td>545.92</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>45.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-137.99</td>
<td>-137.99</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>-45.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-43.08</td>
<td>-43.08</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>29.82</td>
<td>0.44</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>756.76</td>
<td>756.76</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>-29.82</td>
<td>-0.44</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>770.68</td>
<td>770.68</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>-0.25</td>
<td>-0.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-545.92</td>
<td>-545.92</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>0.25</td>
<td>0.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-200.03</td>
<td>-200.03</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>43.08</td>
<td>43.08</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>-0.00</td>
<td>-0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>137.99</td>
<td>137.99</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>106.67</td>
<td>-0.08</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>294.48</td>
<td>294.48</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>-106.67</td>
<td>-0.08</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>-30.23</td>
<td>0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-894.16</td>
<td>-894.16</td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>30.23</td>
<td>0.26</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-158.18</td>
<td>-158.18</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

### EXAMPLE FOR TENSION-ONLY MEMBERS

MEMBER END FORCES  
STRUCTURE TYPE = PLANE

---

**All Units Are -- KNS  MMS  (Local)**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>2</td>
<td>-137.80</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>137.80</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

---

**Application Examples**

**EX. British Design Examples**

**STAAD.Pro**

4812

User Manual
EX. UK-22 Time History Analysis for Sinusoidal Loading

A space frame structure is subjected to a sinusoidal (dynamic) loading. The commands necessary to describe the sine function are demonstrated in this example. Time History analysis is performed on this model.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-22 Time History Analysis for Sinusoidal Loading.STD when you install the program.
STAAD SPACE
*EXAMPLE FOR HARMONIC LOADING GENERATOR

Every STAAD input file has to begin with the word STAAD.
The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

UNIT KNS METER

The units for the data that follows are specified above.

JOINT COORDINATES
1 0 0 0 ; 2 5 0 0 ; 3 5 0 5 ; 4 0 0 5
5 0 7 0 ; 6 2.5 7 0 ; 7 5 7 0 ; 8 5 7 2.5
9 5 7 5 ; 10 2.5 7 5 ; 11 0 7 5
12 0 7 2.5 ; 13 2.5 7 2.5

The joint number followed by the X, Y and Z coordinates are specified above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

MEMBER INCIDENCES
1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12

The members are defined by the joints to which they are connected.

UNIT MMS
MEMBER PROPERTIES

Figure 505: Example Problem No. 22
Members 1 to 16 are defined as PRISmatic sections with width and depth values provided using the YD and ZD options. The UNIT command is specified to change the units for length from METER to MMS.

Joints 1 to 4 are declared to be pinned-supported.

The modulus of elasticity (E), density and Poisson’s ratio are specified following the command CONSTANTS. Built-in default values for concrete are used.

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the sinusoidal loading are provided.

Each data set is individually identified by the number that follows the TYPE command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word FORCE that follows the TYPE 1 command signifies that this data set is for a forcing function. (If you want to specify an earthquake motion, an ACCELERATION may be specified.)

The command FUNCTION SINE indicates that instead of providing the data set as discrete TIME-FORCE pairs, a sinusoidal function, which describes the variation of force with time, is provided.

The parameters of the sine function, such as FREQUENCY, AMPLITUDE, and number of CYCLES of application are then defined. STAAD internally generates discrete TIME-FORCE pairs of data from the sine function in steps of time defined by the default value (see section 5.31.4 of the Technical Reference Manual on page 2630 for more information). The arrival time value indicates the relative value of time at which the force begins to act upon the structure. The modal damping ratio for all the modes is set to 0.075.

The word FORCE that follows the TYPE 1 command signifies that this data set is for a forcing function. (If you want to specify an earthquake motion, an ACCELERATION may be specified.)

The command FUNCTION SINE indicates that instead of providing the data set as discrete TIME-FORCE pairs, a sinusoidal function, which describes the variation of force with time, is provided.

The parameters of the sine function, such as FREQUENCY, AMPLITUDE, and number of CYCLES of application are then defined. STAAD internally generates discrete TIME-FORCE pairs of data from the sine function in steps of time defined by the default value (see section 5.31.4 of the Technical Reference Manual on page 2630 for more information). The arrival time value indicates the relative value of time at which the force begins to act upon the structure. The modal damping ratio for all the modes is set to 0.075.
The above data describe a static load case. A uniformly distributed load of 10 kN/m acting in the negative global Y direction is applied on some members.

```
LOAD 2
  SELFWEIGHT X 1.0
  SELFWEIGHT Y 1.0
  SELFWEIGHT Z 1.0
  JOINT LOAD
  8 12 FX 15.0
  8 12 FY 15.0
  8 12 FZ 15.0
  TIME LOAD
  8 12 FX 1 1
```

This is the second stage of command specification for time history analysis. The two sets of data specified here are:

a. the weights for generation of the mass matrix

b. the application of the time varying loads on the structure.

The weights (from which the masses for the mass matrix are obtained) are specified in the form of selfweight and joint loads.

Following that, the sinusoidal force is applied using the `TIME LOAD` command. The forcing function described by the `TYPE 1` load is applied on joints 8 and 12 and it starts to act starting at a time defined by the 1st arrival time number.

```
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINI
```

The above commands are self explanatory. The `FINISH` command terminates the STAAD run.

### Input File

```
STAAD SPACE EXAMPLE FOR HARMONIC LOADING GENERATOR
UNIT KNS METER
JOINT COORDINATES
  1 0 0 0 ; 2 5 0 0 ; 3 5 0 5 ; 4 0 0 5
  5 0 7 0 ; 6 2.5 7 0 ; 7 5 7 0 ; 8 5 7 2.5
  9 5 7 5 ; 10 2.5 7 5 ; 11 0 7 5
  12 0 7 2.5 ; 13 2.5 7 2.5
MEMBER INCIDENCES
  1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
  7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
  13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12
UNIT MMS
MEMBER PROPERTIES
  1 TO 4 PRIS YD 600 ZD 600
  5 TO 16 PRIS YD 450 ZD 450
SUPPORTS
  1 TO 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
  E 21.0
  POISSON 0.17
  DENSITY 2.36158e-008
  ALPHA 5e-006
  DAMP 0.05
```
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END
DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
DEFINE TIME HISTORY
TYPE 1 FORCE
* FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
FUNCTION SINE
AMPLITUDE 30 FREQUENCY 60 CYCLES 100
*
ARRIVAL TIMES
0.0
DAMPING 0.075
UNIT METER
LOAD 1
MEMBER LOAD
5 6 7 8 9 10 11 12 UNI GY -10.0
LOAD 2
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
8 12 FX 15.0
8 12 FY 15.0
8 12 FZ 15.0
TIME LOAD
8 12 FX 1 1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINI

STAAD Output File

*******************************
*                      *
* STAAD.Pro CONNECT Edition *
* Version 22.01.00.**      *
* Proprietary Program of   *
* Bentley Systems, Inc.    *
* Date= APR 14, 2019       *
* Time= 22:53:47           *
* Licensed to: Bentley Systems Inc *
*******************************

1. STAAD SPACE EXAMPLE FOR HARMONIC LOADING GENERATOR
INPUT FILE: UK-22 Time History Analysis for Sinusoidal Loading.STD
2. UNIT KNS METER
3. JOINT COORDINATES
4. 1 0 0 0 ; 2 5 0 0 ; 3 5 0 5 ; 4 0 0 5
5. 5 0 7 0 ; 6 2 5 7 0 ; 7 5 7 0 ; 8 5 7 2.5
6. 9 5 7 5 ; 10 2 5 7 5 ; 11 0 7 5
7. 12 0 7 2.5 ; 13 2.5 7 2.5
8. MEMBER INCIDENCES
9. 1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
10. 7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
11. 13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12
12. UNIT MMS
13. MEMBER PROPERTIES
14. 1 TO 4 PRIS YD 600 ZD 600
15. 5 TO 16 PRIS YD 450 ZD 450
16. SUPPORTS
17. 1 TO 4 PINNED
18. DEFINE MATERIAL START
19. ISOTROPIC CONCRETE
20. E 21.0
21. POISSON 0.17
22. DENSITY 2.36158E-008
23. ALPHA 5E-006
24. DAMP 0.05
25. G 9.25
26. TYPE CONCRETE
27. STRENGTH FCU 0.0275
28. END DEFINE MATERIAL
29. CONSTANTS
30. MATERIAL CONCRETE ALL
31. DEFINE TIME HISTORY
32. TYPE 1 FORCE
33. * FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
34. FUNCTION SINE
35. AMPLITUDE 30 FREQUENCY 60 CYCLES 100
36. * FOR HARMONIC LOADING GENERATOR -- PAGE NO. 2
37. NUMBER OF POINTS IN DIGITIZED HARMONIC FUNCTION= 1201
38. NUMBER OF POINTS PER QUARTER CYCLE OF HARMONIC FUNCTION= 3
39. FORCE STEP DELTA TIME PER POINT 1.38889E-03
40. ENDING TIME FOR THIS DIGITIZED HARMONIC FUNCTION 1.66667E+00
41. *
42. ARRIVAL TIMES
43. 0.0
44. DAMPING 0.075
45. UNIT METER
46. LOAD 1
47. MEMBER LOAD
48. 5 6 7 8 9 10 11 12 UNI GY -10.0
49. LOAD 2
50. SELFWEIGHT X 1.0
51. SELFWEIGHT Y 1.0
52. SELFWEIGHT Z 1.0
53. JOINT LOAD
54. TOTAL PERFORM ANALYSIS
55. PROBLEM STATISTICS
56. ----------
57. NUMBER OF JOINTS 13 NUMBER OF MEMBERS 16
58. NUMBER OF PLATES 0 NUMBER OF SOLIDS 0
59. NUMBER OF SURFACES 0 NUMBER OF SUPPORTS 4
60. Using 64-bit analysis engine.
61. SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
62. TOTAL PRIMARY LOAD CASES 2, TOTAL DEGREES OF FREEDOM 66
TOTAL LOAD COMBINATION CASES = 0 SO FAR.

***NOTE: MASSES DEFINED UNDER LOAD# 2 WILL FORM THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.

EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 3
EIGEN METHOD : SUBSPACE

----------------------------------------
NUMBER OF MODES REQUESTED = 6
NUMBER OF EXISTING MASSES IN THE MODEL = 27
NUMBER OF MODES THAT WILL BE USED = 6

EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 4
CALCULATED FREQUENCIES FOR LOAD CASE 2

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (CYCLES/SEC)</th>
<th>PERIOD (SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.832</td>
<td>0.54584</td>
</tr>
<tr>
<td>2</td>
<td>1.833</td>
<td>0.54558</td>
</tr>
<tr>
<td>3</td>
<td>2.179</td>
<td>0.45894</td>
</tr>
<tr>
<td>4</td>
<td>18.014</td>
<td>0.05551</td>
</tr>
<tr>
<td>5</td>
<td>18.981</td>
<td>0.05268</td>
</tr>
<tr>
<td>6</td>
<td>23.162</td>
<td>0.04317</td>
</tr>
</tbody>
</table>

MODAL WEIGHT (MODAL MASS TIMES g) IN KNS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.924784E+02</td>
<td>3.675146E-24</td>
<td>1.333421E-21</td>
<td>2.902617E+02</td>
</tr>
<tr>
<td>2</td>
<td>1.338687E-21</td>
<td>3.124512E-30</td>
<td>2.924834E+02</td>
<td>2.907889E+02</td>
</tr>
<tr>
<td>3</td>
<td>1.457481E-25</td>
<td>3.230663E-26</td>
<td>6.769569E-21</td>
<td>4.344889E+02</td>
</tr>
<tr>
<td>4</td>
<td>1.151684E-20</td>
<td>3.984483E-21</td>
<td>1.962648E-19</td>
<td>3.771789E+02</td>
</tr>
<tr>
<td>5</td>
<td>3.567685E-14</td>
<td>1.271663E+02</td>
<td>3.475675E-20</td>
<td>5.768673E+01</td>
</tr>
<tr>
<td>6</td>
<td>8.511521E-03</td>
<td>2.884164E-21</td>
<td>4.373936E-20</td>
<td>6.607558E+01</td>
</tr>
</tbody>
</table>

MASS PARTICIPATION FACTORS

<table>
<thead>
<tr>
<th>MODE</th>
<th>PARTICIPATION FACTORS IN PERCENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SUMM-X   SUMM-Y   SUMM-Z</td>
</tr>
<tr>
<td>1</td>
<td>99.996    0.000     0.000</td>
</tr>
<tr>
<td>2</td>
<td>99.998    43.477    99.998</td>
</tr>
<tr>
<td>3</td>
<td>99.998    43.477    99.998</td>
</tr>
<tr>
<td>4</td>
<td>99.998    43.477    99.998</td>
</tr>
<tr>
<td>5</td>
<td>99.998    43.477    99.998</td>
</tr>
<tr>
<td>6</td>
<td>99.999    43.477    99.998</td>
</tr>
</tbody>
</table>

EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 5

ACTUAL MODAL DAMPING USED IN ANALYSIS

<table>
<thead>
<tr>
<th>MODE</th>
<th>DAMPING</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.07500000</td>
</tr>
<tr>
<td>2</td>
<td>0.07500000</td>
</tr>
<tr>
<td>3</td>
<td>0.07500000</td>
</tr>
<tr>
<td>4</td>
<td>0.07500000</td>
</tr>
<tr>
<td>5</td>
<td>0.07500000</td>
</tr>
<tr>
<td>6</td>
<td>0.07500000</td>
</tr>
</tbody>
</table>

TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED = 6
TIME DURATION OF TIME HISTORY ANALYSIS = 1.667 SECONDS
NUMBER OF TIME STEPS IN THE SOLUTION PROCESS = 1200

55. PRINT ANALYSIS RESULTS

BASE SHEAR UNITS ARE -- KNS METE

MAXIMUM BASE SHEAR X = -1.639967E+00 Y = -2.384186E-07 Z = -9.313226E-10
AT TIMES 0.127778 0.034722 0.063889

ANALYSIS RESULTS

EXAMPLE FOR HARMONIC LOADING GENERATOR -- PAGE NO. 6

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE

------------------------
**Application Examples**

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>X-TRANS</th>
<th>Y-TRANS</th>
<th>Z-TRANS</th>
<th>X-ROTAN</th>
<th>Y-ROTAN</th>
<th>Z-ROTAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
</tr>
</tbody>
</table>

**EXAMPLE FOR HARMONIC LOADING GENERATOR**

---

**SUPPORT REACTIONS -UNIT KNS  METE**

**STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>2.15</td>
<td>50.00</td>
<td>2.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-0.41</td>
<td>-1.13</td>
<td>-0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-2.15</td>
<td>50.00</td>
<td>2.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-0.41</td>
<td>1.13</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-2.15</td>
<td>50.00</td>
<td>-2.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-0.41</td>
<td>-1.13</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**EXAMPLE FOR HARMONIC LOADING GENERATOR**

---

**MEMBER END FORCES**

**STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>50.00</td>
<td>-2.15</td>
<td>2.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-50.00</td>
<td>2.15</td>
<td>-2.15</td>
<td>0.00</td>
<td>-15.03</td>
<td>-15.03</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-1.13</td>
<td>0.41</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.12</td>
<td>2.87</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-50.00</td>
<td>2.15</td>
<td>-2.15</td>
<td>0.00</td>
<td>-15.03</td>
<td>15.03</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.13</td>
<td>0.41</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.12</td>
<td>2.87</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-1.13</td>
<td>-0.41</td>
<td>-0.02</td>
<td>0.00</td>
<td>-0.12</td>
<td>2.87</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>50.00</td>
<td>2.15</td>
<td>-2.15</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>9</td>
<td>-50.00</td>
<td>2.15</td>
<td>-2.15</td>
<td>0.00</td>
<td>15.03</td>
<td>15.03</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.13</td>
<td>-0.41</td>
<td>-0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.12</td>
<td>2.87</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-1.13</td>
<td>-0.41</td>
<td>0.02</td>
<td>0.00</td>
<td>0.12</td>
<td>2.87</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>50.00</td>
<td>-2.15</td>
<td>-2.15</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>-50.00</td>
<td>2.15</td>
<td>2.15</td>
<td>0.00</td>
<td>15.03</td>
<td>-15.03</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**ALL UNITS ARE -- KNS  METE**

**LOCAL**

---

**STAAD.Pro**

4820

**User Manual**
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>4</td>
<td>-1.13</td>
<td>0.41</td>
<td>0.02</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>1.13</td>
<td>-0.41</td>
<td>-0.02</td>
<td>0.00</td>
<td>-0.12</td>
<td>2.87</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>2.09</td>
<td>25.00</td>
<td>0.06</td>
<td>1.34</td>
<td>-0.07</td>
<td>16.37</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-2.09</td>
<td>-0.00</td>
<td>-0.06</td>
<td>-1.34</td>
<td>-0.07</td>
<td>14.88</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>-0.10</td>
<td>-1.03</td>
<td>-0.05</td>
<td>-0.03</td>
<td>0.08</td>
<td>-2.49</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.10</td>
<td>1.03</td>
<td>0.05</td>
<td>0.03</td>
<td>0.06</td>
<td>-0.08</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>6</td>
<td>1.13</td>
<td>-0.41</td>
<td>-0.02</td>
<td>0.00</td>
<td>-0.12</td>
<td>2.87</td>
</tr>
<tr>
<td>11</td>
<td>-2.09</td>
<td>-0.00</td>
<td>-0.06</td>
<td>-1.34</td>
<td>-0.07</td>
<td>14.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>-0.10</td>
<td>-1.03</td>
<td>-0.05</td>
<td>-0.03</td>
<td>0.08</td>
<td>-2.49</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>2.09</td>
<td>25.00</td>
<td>0.06</td>
<td>1.34</td>
<td>-0.07</td>
<td>16.37</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-2.09</td>
<td>0.00</td>
<td>-0.06</td>
<td>-1.34</td>
<td>-0.07</td>
<td>14.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>-0.04</td>
<td>0.14</td>
<td>0.08</td>
<td>-0.38</td>
<td>-0.08</td>
<td>0.14</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.04</td>
<td>-0.14</td>
<td>-0.08</td>
<td>0.38</td>
<td>0.12</td>
<td>-0.21</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>8</td>
<td>-0.04</td>
<td>0.14</td>
<td>-0.08</td>
<td>0.38</td>
<td>0.12</td>
<td>-0.21</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>2.09</td>
<td>25.00</td>
<td>0.06</td>
<td>1.34</td>
<td>-0.07</td>
<td>16.37</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-2.09</td>
<td>0.00</td>
<td>-0.06</td>
<td>-1.34</td>
<td>-0.07</td>
<td>14.88</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**EXAMPLE FOR HARMONIC LOADING GENERATOR**  -- PAGE NO. 9

**MEMBER END FORCES**  STRUCTURE TYPE = SPACE

**----------------------------------------**

**ALL UNITS ARE -- KNS  METE (LOCAL )**

**---------------------------**

**MEMBER END FORCES**  STRUCTURE TYPE = SPACE

**----------------------------------------**

**EXAMPLE FOR HARMONIC LOADING GENERATOR**  -- PAGE NO. 10
EX. UK-23 Spring Support Generation for a Slab on Grade

This example illustrates the usage of commands necessary to automatically generate spring supports for a slab on grade. The slab is subjected to pressure loading and analysis of the structure is performed.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-23 Spring Support Generation for a Slab on Grade.STD when you install the program.

Figure 506: Example Problem No. 23

where:

H = 43m, h1 = 2.83m, h2 = 2.67m, h3 = 2m
V = 13.33m, v1 = 2.17m, v2 = 2m, v3 = 2.61m, v4 = 1.33m

STAAD SPACE SLAB ON GRADE

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The remainder of the words form a title to identify this project.
The units for the data that follows are specified above.

JOINT COORDINATES
1 0.0 0.0 13.33
2 0.0 0.0 12.0
3 0.0 0.0 9.39
4 0.0 0.0 6.78
5 0.0 0.0 4.17
6 0.0 0.0 2.17
7 0.0 0.0 0.0
REPEAT ALL 3 2.83 0.0 0.0
REPEAT 3 2.67 0.0 0.0
REPEAT 5 2 0.0 0.0
REPEAT 3 2.67 0.0 0.0
REPEAT 3 2.83 0.0 0.0

For joints 1 through 7, the joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these seven joints by 2.83 meters, three times. REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7

The incidences of element number 1 is defined and that data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first six elements is then used to generate the incidences of 96 (= 16 x 6) more elements using the REPEAT command.

UNIT CM
ELEMENT PROPERTIES
1 TO 102 TH 14.0

The thickness of elements 1 to 102 is specified as 14 cms following the command ELEMENT PROPERTIES.

UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUB 1570.

The above command is used to instruct STAAD to generate supports with springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade modulus of the soil is specified as
1570 KN/cu.m. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the FY degree of freedom at the joint. See Section 5.27.3 in the STAAD Technical Reference Manual (on page 2517).

PRINT SUPP INFO

This command will enable us to obtain the details of the support conditions which were generated using the earlier commands.

LOAD 1 WEIGHT OF MAT & EARTH
   ELEMENT LOAD
   1 TO 102 PR GY -74.2

The above data describe a static load case. A pressure load of 74.2 kN/sq.m. acting in the negative global Y direction is applied on all the 102 elements.

LOAD 2 'COLUMN LOAD-DL+LL'
   JOINT LOADS
   1 2 FY -965.
   8 9 FY -485.
   5 FY -1373.
   6 FY -2746.
   22 23 FY -1824.
   29 30 FY -912.
   26 FY -2414.
   27 FY -4928.
   43 44 50 51 71 72 78 79 FY -1368.
   47 54 82 FY -2350.
   92 93 FY -912.
   99 100 FY -1824.
   103 FY -2166.
   104 FY -4333.
   113 114 FY -485.
   119 121 FY -965.
   120 121 FY -1216.
   125 FY -2431.

Load case 2 consists of several joint loads acting in the negative global Y direction.

LOADING COMBINATION 101 TOTAL LOAD
   1 1. 2 1.

A load combination case, identified with load case number 101, is specified above. It instructs STAAD to factor loads 1 and 2 by a value of 1.0 and then algebraically add the results.

PERFORM ANALYSIS

The analysis is initiated using the above command.

UNIT CM

LOAD LIST 101
PRINT JOINT DISPLACEMENTS LIST 33 56
PRINT ELEMENT STRESSES LIST 34 67

Joint displacements for joints 33 and 56, and element stresses for elements 34 and 67, for load case 101, is obtained with the help of the above commands.

FINISH

The STAAD run is terminated.
**Input File**

STAAD SPACE SLAB ON GRADE  
UNIT METER KNS  
JOINT COORDINATES  
1 0.0 0.0 13.33  
2 0.0 0.0 12.0  
3 0.0 0.0 9.39  
4 0.0 0.0 6.78  
5 0.0 0.0 4.17  
6 0.0 0.0 2.17  
7 0.0 0.0 0.0  
REPEAT ALL 3 2.83 0.0 0.0  
REPEAT 3 2.67 0.0 0.0  
REPEAT 3 2.67 0.0 0.0  
REPEAT 3 2.83 0.0 0.0  
ELEMENT INCIDENCES  
1 1 8 9 2 TO 6  
REPEAT 16 6 7  
UNIT CM  
ELEMENT PROPERTIES  
1 TO 102 TH 14.0  
UNIT MMS  
DEFINE MATERIAL START  
ISOTROPIC CONCRETE  
E 21.0  
POISSON 0.17  
DENSITY 2.36158e-008  
ALPHA 5e-006  
DAMP 0.05  
G 9.25  
TYPE CONCRETE  
STRENGTH FCU 0.0275  
END DEFINE MATERIAL  
CONSTANTS  
MATERIAL CONCRETE ALL  
UNIT METER  
SUPPORTS  
1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 1570.  
PRINT SUPP INFO  
LOAD 1 'WEIGHT OF MAT & EARTH'  
ELEMENT LOAD  
1 TO 102 PR GY -74.2  
LOAD 2 'COLUMN LOAD-DL+LL'  
JOINT LOADS  
1 2 FY -965.  
8 9 FY -485.  
5 FY -1373.  
6 FY -2746.  
22 23 FY -1824.  
29 30 FY -912.  
26 FY -2414.  
27 FY -4828.  
43 44 50 51 71 72 78 79 FY -1368.  
47 54 82 FY -1175.  
48 55 76 83 FY -2350.
LOADING COMBINATION 101 TOTAL LOAD
1 1 2 1.
PERFORM ANALYSIS
UNIT CM
LOAD LIST 101
PRINT JOINT DISPLACEMENTS LIST 33 56
PRINT ELEMENT STRESSES LIST 34 67
FINISH

STAAD Output File

****************************************************
**                                                  **
**           STAAD.Pro CONNECT Edition              **
**           Version 22.01.00.**                    **
**           Proprietary Program of                **
**           Bentley Systems, Inc.                 **
**           Date= APR 14, 2019                    **
**           Time= 22:53:52                        **
**                                                  **
**  Licensed to: Bentley Systems Inc               **
****************************************************

1. STAAD SPACE SLAB ON GRADE
INPUT FILE: UK-23 Spring Support Generation for a Slab on Grade.STD
2. UNIT METER KNS
3. JOINT COORDINATES
4. 1 0.0 0.0 13.33
5. 2 0.0 0.0 12.0
6. 3 0.0 0.0 9.39
7. 4 0.0 0.0 6.78
8. 5 0.0 0.0 4.17
9. 6 0.0 0.0 2.17
10. 7 0.0 0.0 0.0
11. REPEAT ALL 3 2.83 0.0 0.0
12. REPEAT 3 2.67 0.0 0.0
13. REPEAT 5 2 0.0 0.0
14. REPEAT 3 2.67 0.0 0.0
15. REPEAT 3 2.83 0.0 0.0
16. ELEMENT INCIDENCES
17. 1 1 8 9 2 TO 6
18. REPEAT 16 6 7
19. UNIT CM
20. ELEMENT PROPERTIES
21. 1 TO 102 TH 14.0
22. UNIT MMS
23. DEFINE MATERIAL START
24. ISOTROPIC CONCRETE
25. E 21.0
26. POISSON 0.17
27. DENSITY 2.36158E-008
28. ALPHA 5E-006
29. DAMP 0.05
30. G 9.25
31. TYPE CONCRETE
32. STRENGTH FCU 0.0275
33. END DEFINE MATERIAL
34. CONSTANTS
35. MATERIAL CONCRETE ALL
36. UNIT METER
37. SUPPORTS
38. 1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 1570.
    SLAB ON GRADE

39. PRINT SUPP INFO
SUPP INFO
SUPPORT INFORMATION (1=FIXED, 0=RELEASED)

<table>
<thead>
<tr>
<th>JOINT</th>
<th>FORCE-X/</th>
<th>FORCE-Y/</th>
<th>FORCE-Z/</th>
<th>MOM-X/</th>
<th>MOM-Y/</th>
<th>MOM-Z/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>KFX</td>
<td>KFY</td>
<td>KFZ</td>
<td>KMX</td>
<td>KMY</td>
<td>KMZ</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

---

STAAD.Pro 4827 User Manual
### Application Examples

#### EX. British Design Examples

<table>
<thead>
<tr>
<th>No.</th>
<th>1</th>
<th>0</th>
<th>1</th>
<th>0</th>
<th>1</th>
<th>0</th>
<th>0.0</th>
<th>9263.9</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>SLAB ON GRADE</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>4820.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>21</td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2871.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>22</td>
</tr>
<tr>
<td>22</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8505.5</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>23</td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>11268.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>24</td>
</tr>
<tr>
<td>24</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>11268.7</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>25</td>
</tr>
<tr>
<td>25</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9951.8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>26</td>
</tr>
<tr>
<td>26</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9002.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>27</td>
</tr>
<tr>
<td>27</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>4684.5</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>28</td>
</tr>
<tr>
<td>28</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2787.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>29</td>
</tr>
<tr>
<td>29</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8258.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>30</td>
</tr>
<tr>
<td>30</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>10940.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>31</td>
</tr>
<tr>
<td>31</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9662.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>32</td>
</tr>
<tr>
<td>32</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8740.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>33</td>
</tr>
<tr>
<td>33</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>4548.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>34</td>
</tr>
<tr>
<td>34</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2787.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>35</td>
</tr>
<tr>
<td>35</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8258.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>36</td>
</tr>
<tr>
<td>36</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>10940.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>37</td>
</tr>
<tr>
<td>37</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9662.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>38</td>
</tr>
<tr>
<td>38</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8740.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>39</td>
</tr>
<tr>
<td>39</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>4548.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>40</td>
</tr>
<tr>
<td>40</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2787.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>41</td>
</tr>
<tr>
<td>41</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8258.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>42</td>
</tr>
<tr>
<td>42</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>10940.9</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>43</td>
</tr>
<tr>
<td>43</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9662.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>44</td>
</tr>
<tr>
<td>44</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8740.1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>45</td>
</tr>
<tr>
<td>45</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>4548.2</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>46</td>
</tr>
<tr>
<td>46</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2787.6</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>47</td>
</tr>
<tr>
<td>47</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8258.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>48</td>
</tr>
<tr>
<td>48</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8450.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>49</td>
</tr>
</tbody>
</table>

---

**STAAD.Pro User Manual**

Version 4828
Application Examples
EX. British Design Examples

<table>
<thead>
<tr>
<th></th>
<th>SLAB ON GRADE</th>
</tr>
</thead>
<tbody>
<tr>
<td>49</td>
<td>0.0 7643.5 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>50</td>
<td>0.0 3977.6 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>51</td>
<td>0.0 2088.1 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>52</td>
<td>0.0 6185.8 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>53</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>54</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>55</td>
<td>0.0 7237.7 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>56</td>
<td>0.0 6546.9 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>57</td>
<td>0.0 3406.9 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>58</td>
<td>0.0 2088.1 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>59</td>
<td>0.0 6185.8 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>60</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>61</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>62</td>
<td>0.0 7237.7 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>63</td>
<td>0.0 6546.9 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>64</td>
<td>0.0 3406.9 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>65</td>
<td>0.0 2088.1 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>66</td>
<td>0.0 6185.8 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>67</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>68</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>69</td>
<td>0.0 7237.7 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>70</td>
<td>0.0 6546.9 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>71</td>
<td>0.0 3406.9 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>72</td>
<td>0.0 2088.1 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>73</td>
<td>0.0 6185.8 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>74</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>75</td>
<td>0.0 8195.4 0.0 0.0 0.0 0.0 0.0</td>
</tr>
<tr>
<td>76</td>
<td>0.0 7237.7 0.0 0.0 0.0 0.0 0.0</td>
</tr>
</tbody>
</table>

STAAD.Pro 4829 User Manual
### Application Examples

#### EX. British Design Examples

<table>
<thead>
<tr>
<th>77</th>
<th>1</th>
<th>0</th>
<th>0.0</th>
<th>3406.9</th>
<th>1</th>
<th>0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>78</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2437.9</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

SLAB ON GRADE

--- PAGE NO. 5

<table>
<thead>
<tr>
<th>79</th>
<th>1</th>
<th>0</th>
<th>0.0</th>
<th>7221.9</th>
<th>1</th>
<th>0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
<th>0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>80</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9568.1</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>81</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9568.1</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>82</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8450.0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>83</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>7643.5</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>84</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>3977.6</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>85</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2787.6</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>86</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8258.0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>87</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>10940.9</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>88</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>10940.9</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>89</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9662.3</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>90</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8740.1</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>91</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>4548.2</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>92</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2787.6</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>93</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8258.0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>94</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>10940.9</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>95</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>10940.9</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>96</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9662.3</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>97</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8740.1</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>98</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>4548.2</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>99</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2871.1</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>100</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>8505.5</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>101</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>11268.7</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>102</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>11268.7</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>103</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9951.8</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>104</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>9002.0</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>105</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>2871.1</td>
<td>1</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. British Design Examples**

```
0.0 4684.5 0.0 0.0 0.0 0.0
106 1 0 1 0 1 0
0.0 2954.7 0.0 0.0 0.0 0.0
SLAB ON GRADE
107 1 0 1 0 1 0
0.0 8752.9 0.0 0.0 0.0 0.0
108 1 0 1 0 1 0
0.0 11596.5 0.0 0.0 0.0 0.0
109 1 0 1 0 1 0
0.0 11596.5 0.0 0.0 0.0 0.0
110 1 0 1 0 1 0
0.0 10241.3 0.0 0.0 0.0 0.0
111 1 0 1 0 1 0
0.0 9263.9 0.0 0.0 0.0 0.0
112 1 0 1 0 1 0
0.0 4820.8 0.0 0.0 0.0 0.0
113 1 0 1 0 1 0
0.0 2954.7 0.0 0.0 0.0 0.0
114 1 0 1 0 1 0
0.0 8752.9 0.0 0.0 0.0 0.0
115 1 0 1 0 1 0
0.0 11596.5 0.0 0.0 0.0 0.0
116 1 0 1 0 1 0
0.0 11596.5 0.0 0.0 0.0 0.0
117 1 0 1 0 1 0
0.0 10241.3 0.0 0.0 0.0 0.0
118 1 0 1 0 1 0
0.0 9263.9 0.0 0.0 0.0 0.0
119 1 0 1 0 1 0
0.0 4820.8 0.0 0.0 0.0 0.0
120 1 0 1 0 1 0
0.0 1477.3 0.0 0.0 0.0 0.0
121 1 0 1 0 1 0
0.0 4376.5 0.0 0.0 0.0 0.0
122 1 0 1 0 1 0
0.0 5798.2 0.0 0.0 0.0 0.0
123 1 0 1 0 1 0
0.0 5798.2 0.0 0.0 0.0 0.0
124 1 0 1 0 1 0
0.0 5120.7 0.0 0.0 0.0 0.0
125 1 0 1 0 1 0
0.0 4631.9 0.0 0.0 0.0 0.0
126 1 0 1 0 1 0
0.0 2410.4 0.0 0.0 0.0 0.0
*************** END OF DATA FROM INTERNAL STORAGE ***************
40. LOAD 1 'WEIGHT OF MAT & EARTH'
41. ELEMENT LOAD
42. 1 TO 102 PR GY -74.2
43. LOAD 2 'COLUMN LOAD-DL+LL'
44. JOINT LOADS
45. 1 2 FY -965.
46. 8 9 FY -485.
47. 5 FY -1373.
48. 6 FY -2746.
SLAB ON GRADE
49. 22 23 FY -1824.
50. 29 30 FY -912.
51. 26 FY -2414.
```
52. 27    FY -4828.
53. 43 44 50 51 71 72 78 79 FY -1368.
54. 47 54 82    FY -1175.
55. 48 55 76 83 FY -2350.
56. 92 93    FY -912.
57. 99 100    FY -1824.
58. 103    FY -2166.
59. 104    FY -4333.
60. 113 114    FY -485.
61. 120 121    FY -965.
62. 124    FY -1216.
63. 125    FY -2431.
64. LOADING COMBINATION 101 TOTAL LOAD
65. 1 1. 2 1.
66. PERFORM ANALYSIS

PROBLEM STATISTICS
---------------------------------
NUMBER OF JOINTS        126  NUMBER OF MEMBERS       0
NUMBER OF PLATES        102  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS    126

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     378
TOTAL LOAD COMBINATION CASES =     1  SO FAR.
67. UNIT CM
68. LOAD LIST 101
69. PRINT JOINT DISPLACEMENTS LIST 33 56
JOINT DISPLACE LIST 33
SLAB ON GRADE

JOINT DISPLACEMENT (CM RADIANS)  STRUCTURE TYPE = SPACE

------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
33  101     0.0000  -10.5679    0.0000   -0.0259    0.0000    0.0552
56  101     0.0000  -12.1989    0.0000    0.0660    0.0000    0.0288

************** END OF LATEST ANALYSIS RESULT **************
70. PRINT ELEMENT STRESSES LIST 34 67
ELEMENT STRESSES LIST 34
SLAB ON GRADE

ELEMENT STRESSES  FORCE,LENGTH UNITS= KNS CM

STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

----------------
ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
VONT       VONB         SX          SY          SXY
TRESCAT    TRESCAB
34    101        -0.02       -0.03        2.22       17.75       41.14
2.24        2.24        0.00        0.00        0.00
2.56        2.56
5.47        5.47
36    101         0.28        0.03       62.65       39.76       88.62
4.99        4.99        0.00        0.00        0.00
5.47        5.47
70    101        -0.02       -0.03        2.22       17.75       41.14
2.24        2.24        0.00        0.00        0.00
2.56        2.56
5.47        5.47

TOP : SMAX=       1.59 SMIN=      -0.98 TMAX=       1.28 ANGLE= 50.3
BOT: SMAX=       -1.59 SMIN=      -0.98 TMAX=       1.28 ANGLE=-39.7
BOTT: SMAX=       -1.59 SMIN=      -0.98 TMAX=       1.28 ANGLE=-39.7
67    101         0.28        0.03       62.65       39.76       88.62
4.99        4.99        0.00        0.00        0.00
5.47        5.47

BOTT: SMAX=       -1.17 TMAX=       2.74 ANGLE= 41.3
BOTT: SMAX=       -4.30 TMAX=       2.74 ANGLE=-48.7

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
MAXIMUM  MINIMUM  MAXIMUM  MAXIMUM  MAXIMUM
PRINCIPAL  PRINCIPAL  SHEAR  VONMISES  TRESCA
STRESS  STRESS  STRESS  STRESS  STRESS
EX. UK-24 Analysis of a Concrete Block Using Solid Elements

This is an example of the analysis of a structure modeled using solid finite elements. This example also illustrates the method for applying an enforced displacement on the structure.

This problem is installed with the program by default to C: \Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-24 Analysis of a Concrete Block Using Solid Elements.STD when you install the program.
STAAD SPACE

*EXAMPLE PROBLEM USING SOLID ELEMENTS

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

UNIT KNS MET

The units for the data that follows are specified above.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1  0.0  0.0  2.0  4  0.0  3.0  2.0</td>
</tr>
<tr>
<td>5  1.0  0.0  2.0  8  1.0  3.0  2.0</td>
</tr>
<tr>
<td>9  2.0  0.0  2.0 12  2.0  3.0  2.0</td>
</tr>
<tr>
<td>21  0.0  0.0  1.0 24  0.0  3.0  1.0</td>
</tr>
<tr>
<td>25  1.0  0.0  1.0 28  1.0  3.0  1.0</td>
</tr>
<tr>
<td>29  2.0  0.0  1.0 32  2.0  3.0  1.0</td>
</tr>
<tr>
<td>41  0.0  0.0  0.0 44  0.0  3.0  0.0</td>
</tr>
<tr>
<td>45  1.0  0.0  0.0 48  1.0  3.0  0.0</td>
</tr>
<tr>
<td>49  2.0  0.0  0.0 52  2.0  3.0  0.0</td>
</tr>
</tbody>
</table>

The joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of some of those nodes are generated utilizing the fact that they are equally spaced between the extremities.

<table>
<thead>
<tr>
<th>ELEMENT INCIDENCES SOLID</th>
</tr>
</thead>
<tbody>
<tr>
<td>1  1  5  6  2  21  25  26  22  T0  3</td>
</tr>
</tbody>
</table>
The incidences of solid elements are defined above. The word SOLID is used to signify that these are 8-node solid elements as opposed to 3-noded or 4-noded plate elements. Each line contains the data for generating 3 elements. For example, element number 1 is first defined by all of its 8 nodes. Then, increments of 1 to the joint number and 1 to the element number (the defaults) are used for generating incidences for elements 2 and 3. Similarly, incidences of elements 4, 7 and 10 are defined while those of 5, 6, 8, 9, 11 and 12 are generated.

UNIT MMS
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 210
  POISSON 0.25
  DENSITY 7.5e-008
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METERL

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

PRINT ELEMENT INFO SOLID LIST 1 TO 5

This command will enable us to obtain, in a tabular form, the details of the incidences and material property values of elements 1 to 5.

SUPPORTS
  1 5 21 25 29 41 45 49 PINNED
  9 ENFORCED

The above lines contain the data for supports for the model. The ENFORCED support condition is used to declare a point at which an enforced displacement load is applied later (see load case 3).

LOAD 1
  SELF Y -1.0
  JOINT LOAD
  28 FY -1000.0

The above data describe a static load case. It consists of selfweight loading and a joint load, both in the negative global Y direction.

LOAD 2
  JOINT LOADS
  2 TO 4 22 TO 24 42 TO 44 FX 100.0

Load case 2 consists of several joint loads acting in the positive global X direction.

LOAD 3
  SUPPORT DISPLACEMENT
  9 FX 0.0011
Load case 3 consists of an enforced displacement along the global X direction at node 9. The displacement in the other enforced support degrees of freedom will default to zero.

```
UNIT POUND FEET
LOAD 4
ELEMENT LOAD SOLIDS
  3 6 9 12 FACE 4 PRE GY -500.0
```

In Load case 4, a pressure load of 500 pounds/sq.ft is applied on Face # 4 of solid elements 3, 6, 9 and 12. Face 4 is defined as shown in the following table:

<table>
<thead>
<tr>
<th>Face Number</th>
<th>Surface Joints</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>f1</td>
</tr>
<tr>
<td>1 front</td>
<td>Jt 1</td>
</tr>
<tr>
<td>2 bottom</td>
<td>Jt 1</td>
</tr>
<tr>
<td>3 left</td>
<td>Jt 1</td>
</tr>
<tr>
<td>4 top</td>
<td>Jt 4</td>
</tr>
<tr>
<td>5 right</td>
<td>Jt 2</td>
</tr>
<tr>
<td>6 back</td>
<td>Jt 5</td>
</tr>
</tbody>
</table>

The above table, and other details of this type of loading can be found in section 5.32.3.2 of the STAAD.Pro Technical Reference manual (on page 2660).

```
UNIT KNS MMS
LOAD 5
REPEAT LOAD
  1 1.0 2 1.0 3 1.0 4 1.0
```

Load case 5 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We want the program to analyze the structure for loads from cases 1 through 4 acting simultaneously. In other words, the above instruction is the same as the following:

```
LOAD 5
SELF Y -1.0
JOINT LOAD
  28 FY -1000.0
  2 TO 4 22 TO 24 42 TO 44 FX 100.0
SUPPORT DISPLACEMENT
  9 FX .0011
ELEMENT LOAD SOLIDS
  3 6 9 12 FACE 4 PRE GY -500.0
LOAD COMB 10
  1 1.0 2 1.0
```

Load case 10 is a combination load case, which combines the effects of cases 1 & 2. While the syntax of this might look very similar to that of the REPEAT LOAD case shown in case 5, there is a fundamental difference. In a REPEAT LOAD case, the program computes the displacements by multiplying the inverted stiffness matrix by the...
load vector built for the REPEAT LOAD case. But in solving load combination cases, the program merely calculates the end results (displacements, forces, reactions) by gathering up the corresponding values from the individual components of the combination case, factoring them, and then algebraically summing them up. This difference in approach is quite important in that non-linear problems such as PDELTA ANALYSIS, MEMBER TENSION, and MEMBER COMPRESSION situations, changes in support conditions etc. should be handled using REPEAT LOAD cases, not load combination cases.

PERFORM ANALYSIS PRINT STATICS CHECK

A static equilibrium report, consisting of total applied loading and total support reactions from each primary load case is requested along with the instructions to carry out a linear static analysis.

PRINT JOINT DISPLACEMENTS LIST 8 9

Global displacements at nodes 8 and 9 are obtained using the above command.

UNIT KNS METER
PRINT SUPPORT REACTIONS

Reactions at the supports are obtained using the above command.

UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6

This command requests the program to provide the element stress results at the nodes of elements 4 and 6. The results will be printed for all the load cases. The word SOLID is used to signify that these are solid elements as opposed to plate or shell elements.

FINISH

The STAAD run is terminated.

Input File

STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
UNIT KNS MET
JOINT COORDINATES
1 0.0 0.0 2.0 4 0.0 3.0 2.0
5 1.0 0.0 2.0 8 1.0 3.0 2.0
9 2.0 0.0 2.0 12 2.0 3.0 2.0
21 0.0 0.0 1.0 24 0.0 3.0 1.0
25 1.0 0.0 1.0 28 1.0 3.0 1.0
29 2.0 0.0 1.0 32 2.0 3.0 1.0
41 0.0 0.0 0.0 44 0.0 3.0 0.0
45 1.0 0.0 0.0 48 1.0 3.0 0.0
49 2.0 0.0 0.0 52 2.0 3.0 0.0
ELEMENT INCIDENCES SOLID
1 1 5 6 2 21 25 26 22 TO 3
4 21 25 26 22 41 45 46 42 TO 6 1 1
7 5 9 10 6 25 29 30 26 TO 9 1 1
10 25 29 30 26 45 49 50 46 TO 12 1 1
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.25
DENSITY 7.5e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
PRINT ELEMENT INFO SOLID LIST 1 TO 5
SUPPORTS
1 5 21 25 29 41 45 49 PINNED
9 ENFORCED BUT MX MY MZ
LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
LOAD 2
JOINT LOADS
2 TO 4 22 TO 24 42 TO 44 FX 100.0
LOAD 3
SUPPORT DISPLACEMENT
9 FX .0001
UNIT POUND FEET
LOAD 4
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
UNIT KNS MMS
LOAD 5
REPEAT LOAD
1 1.0 2 1.0 3 1.0 4 1.0
LOAD COMB 10
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STAT CHECK
PRINT JOINT DISPLACEMENTS LIST 8 9
UNIT KNS METER
PRINT SUPPORT REACTIONS
UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
FINISH

STAAD Output File

******************************************************************************
*  Licensed to: Bentley Systems Inc                                         *
******************************************************************************
1. STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
INPUT FILE: UK-24 Analysis of a Concrete Block Using Solid Elements.STD
2. UNIT KNS MET
3. JOINT COORDINATES
4. 1 0.0 0.0 2.0 4 0.0 3.0 2.0
5. 5 1.0 0.0 2.0 8 1.0 3.0 2.0
6. 9 2.0 0.0 2.0 12 2.0 3.0 2.0
ELEMENT INCIDENCES SOLID
1. 1 5 6 2 21 25 26 22 TO 3
2. 7 5 10 6 25 29 30 26 TO 9 11
3. 10 25 29 30 26 45 49 50 46 TO 12 11

DEFINE MATERIAL START
E 210
POISSON 0.25
DENSITY 7.5E-008
ALPHA 6E-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL

DEFINE MATERIAL ALL
UNIT METER
PRINT ELEMENT INFO SOLID LIST 1 TO 5
ELEMENT INFO SOLID LIST

EXAMPLE PROBLEM USING SOLID ELEMENTS -- PAGE NO. 2

ELEMENT NODE-1 NODE-2 NODE-3 NODE-4 NODE-5 NODE-6 NODE-7 NODE-8
1 1 5 6 2 21 25 26 22
2 2 6 7 3 22 26 27 23
3 3 7 8 4 23 27 28 24
4 21 25 26 22 41 45 46 42
5 22 26 27 23 42 46 47 43

EXAMPLE PROBLEM USING SOLID ELEMENTS -- PAGE NO. 3

MATERIAL PROPERTIES.
-------------------
ALL UNITS ARE - KNS METE

ELEMENT YOUNG'S MODULUS MODULUS OF RIGIDITY DENSITY ALPHA
1 2.1000002E+08 0.0000000E+00 7.5000E+01 6.0000E-06
2 2.1000002E+08 0.0000000E+00 7.5000E+01 6.0000E-06
3 2.1000002E+08 0.0000000E+00 7.5000E+01 6.0000E-06
4 2.1000002E+08 0.0000000E+00 7.5000E+01 6.0000E-06
5 2.1000002E+08 0.0000000E+00 7.5000E+01 6.0000E-06

SUPPORTS
1 5 21 25 29 41 45 49 PINNED
9 ENFORCED BUT MX MY MZ
LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
LOAD 2
JOINT LOADS
2 TO 4 22 TO 24 42 TO 44 FX 100.0
LOAD 3
SUPPORT DISPLACEMENT
9 FX .0011
UNIT POUND FEET

STAAD.Pro 4839 User Manual
LOAD 4
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
UNIT KNS MMS
LOAD 5
REPEAT LOAD
1 1.0 2 1.0 3 1.0 4 1.0
LOAD COMB 10
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STAT CHECK

PROBLEM STATISTICS

NUMBER OF JOINTS 36
NUMBER OF MEMBERS 0
NUMBER OF PLATES 0
NUMBER OF SOLIDS 12
NUMBER OF SURFACES 0
NUMBER OF SUPPORTS 9

EXAMPLE PROBLEM USING SOLID ELEMENTS

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES = 5, TOTAL DEGREES OF FREEDOM = 84
TOTAL LOAD COMBINATION CASES = 1 SO FAR.

EXAMPLE PROBLEM USING SOLID ELEMENTS

TOTAL APPLIED LOAD 1
***TOTAL APPLIED LOAD (KNS MMS) SUMMARY (LOADING 1)
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -1900.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 1900000.15  MY= 0.00  MZ= -1900000.15

TOTAL REACTION LOAD 1
***TOTAL REACTION LOAD (KNS MMS) SUMMARY (LOADING 1)
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 1900.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -1900000.15  MY= -0.00  MZ= 1900000.15

MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 1)
MAXIMUMS AT NODE
X = -1.21106E-04  Y = -1.15439E-03  Z = -1.21106E-04
RX= 0.00000E+00  RY= 0.00000E+00  RZ= 0.00000E+00

EXAMPLE PROBLEM USING SOLID ELEMENTS

TOTAL APPLIED LOAD 2
***TOTAL APPLIED LOAD (KNS MMS) SUMMARY (LOADING 2)
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 0.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.00  MY= 0.00  MZ= 0.00

MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 2)
MAXIMUMS AT NODE
X = -1.15439E-03  Y = -1.15439E-03  Z = 1.21106E-04
RX= 0.00000E+00  RY= 0.00000E+00  RZ= 0.00000E+00

EXAMPLE PROBLEM USING SOLID ELEMENTS

TOTAL APPLIED LOAD 2
***TOTAL APPLIED LOAD (KNS MMS) SUMMARY (LOADING 2)
SUMMATION FORCE-X = 900.00
SUMMATION FORCE-Y = 0.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.00  MY= 900000.03  MZ= -1800000.06
TOTAL REACTION LOAD 2

***TOTAL REACTION LOAD ( KNS MMS ) SUMMARY (LOADING 2 )
SUMMATION FORCE-X = -900.00
SUMMATION FORCE-Y = -0.00
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.00  MY= -900000.03  MZ= 1800000.06
MAXIMUM DISPLACEMENTS ( CM /RADIANS ) (LOADING 2)
MAXIMUMS AT NODE
X = 2.22892E-03  4
Y = 7.83934E-04  44
Z = 9.49033E-05  10
RX= 0.00000E+00  0
RY= 0.00000E+00  0
RZ= 0.00000E+00  0

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 3
TOTAL APPLIED LOAD

***TOTAL APPLIED LOAD ( KNS MMS ) SUMMARY (LOADING 3 )
SUMMATION FORCE-X = 0.0000000E+00
SUMMATION FORCE-Y = 0.0000000E+00
SUMMATION FORCE-Z = 0.0000000E+00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 0.0000000E+00  MY= 0.0000000E+00  MZ= 0.0000000E+00
TOTAL REACTION LOAD 3

***TOTAL REACTION LOAD( KNS MMS ) SUMMARY (LOADING 3 )
SUMMATION FORCE-X = 2.4273804E-11
SUMMATION FORCE-Y = 3.0342255E-12
SUMMATION FORCE-Z = 3.0342255E-12
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 4.9324371E-08  MY= 4.6036080E-08  MZ= 3.2882914E-09
EXAMPLE PROBLEM USING SOLID ELEMENTS -- PAGE NO. 7
MAXIMUM DISPLACEMENTS ( CM /RADIANS ) (LOADING 3)
MAXIMUMS AT NODE
X = 1.10000E-01  9
Y = -1.21497E-02  6
Z = 1.61372E-02  24
RX= 0.00000E+00  0
RY= 0.00000E+00  0
RZ= 0.00000E+00  0

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 4
CENTER OF FORCE BASED ON Y FORCES ONLY (MMS ).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.999999993E+03
Y = 0.299999998E+04
Z = 0.999999993E+03
TOTAL APPLIED LOAD 4

***TOTAL APPLIED LOAD ( KNS MMS ) SUMMARY (LOADING 4 )
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -95.76
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 95760.52  MY= 0.00  MZ= -95760.52
TOTAL REACTION LOAD 4
***TOTAL REACTION LOAD (KNS MMS) SUMMARY (LOADING 4)
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 95.76
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX = -95760.52 MY = 0.00 MZ = 95760.52

MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 4)
MAXIMUMS AT NODE
X = 3.17652E-06 50
Y = -3.35288E-05 28
Z = -3.17652E-06 50
RX = 0.00000E+00 0
RY = 0.00000E+00 0
RZ = 0.00000E+00 0

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 5
EXAMPLE PROBLEM USING SOLID ELEMENTS -- PAGE NO. 8
CENTER OF FORCE BASED ON X FORCES ONLY (MMS).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.000000000E+00
Y = 0.199999999E+04
Z = 0.999999993E+03

CENTER OF FORCE BASED ON Y FORCES ONLY (MMS).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.999999993E+03
Y = 0.232356609E+04
Z = 0.999999993E+03

TOTAL APPLIED LOAD 5
***TOTAL APPLIED LOAD (KNS MMS) SUMMARY (LOADING 5)
SUMMATION FORCE-X = 900.00
SUMMATION FORCE-Y = -1995.76
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX = 1995760.67 MY = 900000.03 MZ = -3795760.73

TOTAL REACTION LOAD 5
***TOTAL REACTION LOAD (KNS MMS) SUMMARY (LOADING 5)
SUMMATION FORCE-X = -900.00
SUMMATION FORCE-Y = 1995.76
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX = -1995760.67 MY = -900000.03 MZ = 3795760.73

MAXIMUM DISPLACEMENTS (CM/RADIANS) (LOADING 5)
MAXIMUMS AT NODE
X = 1.10000E-01 9
Y = -1.23568E-02 6
Z = 1.61372E-02 24
RX = 0.00000E+00 0
RY = 0.00000E+00 0
RZ = 0.00000E+00 0

************** END OF DATA FROM INTERNAL STORAGE **************

57. PRINT JOINT DISPLACEMENTS LIST 8 9

JOINT DISPLACE LIST 8
EXAMPLE PROBLEM USING SOLID ELEMENTS -- PAGE NO. 9

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE

---------------------

JOINT LOAD X-TRANS Y-TRANS Z-TRANS X-ROTAN Y-ROTAN Z-ROTAN
8 1 0.0000 -0.0002 -0.0001 0.0000 0.0000 0.0000
2 0.0020 0.0000 -0.0000 0.0000 0.0000 0.0000
3 0.0193 -0.0049 0.0089 0.0000 0.0000 0.0000

STAAD.Pro 4842 User Manual
### SUPPORT REACTION EXAMPLE PROBLEM USING SOLID ELEMENTS -- PAGE NO. 10

**SUPPORT REACTIONS - UNIT KNS METER**

**STRUCTURE TYPE = SPACE**

<table>
<thead>
<tr>
<th>JOINT</th>
<th>LOAD</th>
<th>FORCE-X</th>
<th>FORCE-Y</th>
<th>FORCE-Z</th>
<th>MOM-X</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>27.47</td>
<td>128.97</td>
<td>-27.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-72.24</td>
<td>-232.67</td>
<td>42.18</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-2022.70</td>
<td>-302.04</td>
<td>-1192.39</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1.52</td>
<td>6.63</td>
<td>-1.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-2065.94</td>
<td>-399.11</td>
<td>-1179.21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-44.76</td>
<td>-103.70</td>
<td>14.70</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.00</td>
<td>236.52</td>
<td>-54.44</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-62.32</td>
<td>11.42</td>
<td>-0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-16410.02</td>
<td>7434.80</td>
<td>-2287.95</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>11.97</td>
<td>-2.98</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-16472.33</td>
<td>7694.71</td>
<td>-2345.41</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-62.32</td>
<td>247.94</td>
<td>-54.49</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>54.44</td>
<td>236.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-159.92</td>
<td>-450.84</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-3341.67</td>
<td>-2923.60</td>
<td>-1877.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>2.98</td>
<td>11.97</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-3444.18</td>
<td>-3125.95</td>
<td>-1877.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-105.49</td>
<td>-214.32</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>25</td>
<td>1</td>
<td>-8.00</td>
<td>438.86</td>
<td>-8.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-138.00</td>
<td>9.51</td>
<td>-8.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-19197.98</td>
<td>5248.66</td>
<td>-10975.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>21.34</td>
<td>21.34</td>
<td>-8.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-19335.98</td>
<td>5717.57</td>
<td>-10975.25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-138.00</td>
<td>-447.56</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>29</td>
<td>1</td>
<td>-54.44</td>
<td>236.52</td>
<td>8.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-170.27</td>
<td>431.34</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>3902.73</td>
<td>512.05</td>
<td>3842.64</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-2.98</td>
<td>11.97</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>3675.05</td>
<td>1191.87</td>
<td>3842.64</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-224.70</td>
<td>667.85</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>41</td>
<td>1</td>
<td>27.47</td>
<td>128.97</td>
<td>27.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-72.24</td>
<td>-232.67</td>
<td>-42.18</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-891.15</td>
<td>-2739.86</td>
<td>-1598.54</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>1.52</td>
<td>6.63</td>
<td>1.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-934.39</td>
<td>-2836.93</td>
<td>-1611.72</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>10</td>
<td>-44.76</td>
<td>-103.70</td>
<td>-14.70</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>45</td>
<td>1</td>
<td>-0.00</td>
<td>236.52</td>
<td>54.44</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-62.32</td>
<td>11.42</td>
<td>0.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-430.44</td>
<td>-752.46</td>
<td>-237.57</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>-0.00</td>
<td>11.97</td>
<td>2.98</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>-492.75</td>
<td>-492.56</td>
<td>-180.10</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>JOINT LOAD</td>
<td>FORCE-X</td>
<td>FORCE-Y</td>
<td>FORCE-Z</td>
<td>MOM-X</td>
<td>MOM-Y</td>
<td>MOM Z</td>
<td></td>
</tr>
<tr>
<td>------------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
<td>-------</td>
<td>-------</td>
<td>-------</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>108.83</td>
<td>355.21</td>
<td>72.50</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>49</td>
<td>-27.47</td>
<td>128.97</td>
<td>-27.47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-1.52</td>
<td>6.63</td>
<td>1.52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

**EXAMPLE PROBLEM USING SOLID ELEMENTS**  
**-- PAGE NO. 11**

**SUPPORT REACTIONS -UNIT KNS**

**STRUCTURE TYPE = SPACE**

**-------------------**

**JOINT LOAD**  
**FORCE-X**  
**FORCE-Y**  
**FORCE-Z**  
**MOM-X**  
**MOM-Y**  
**MOM Z**

5  
-888.61  
2435.62  
1266.41  
0.00  
0.00  
0.00

9  
1  
-27.47  
128.97  
-27.47  
0.00  
0.00  
0.00

2  
-81.35  
226.24  
-45.03  
0.00  
0.00  
0.00

3  
39169.49  
-8551.31  
13133.66  
0.00  
0.00  
0.00

4  
-1.52  
6.63  
-1.52  
0.00  
0.00  
0.00

5  
39059.14  
-8189.46  
13059.64  
0.00  
0.00  
0.00

10  
-108.83  
355.21  
-72.50  
0.00  
0.00  
0.00

*************** END OF LATEST ANALYSIS RESULT ***************

60. **UNIT NEWTON MMS**

61. **PRINT ELEMENT JOINT STRESS SOLID LIST 4 6**

**ELEMENT JOINT STRESS SOLID**

**EXAMPLE PROBLEM USING SOLID ELEMENTS**  
**-- PAGE NO. 12**

**ELEMENT STRESSES**  
**UNITS= NEWTMMS**

<table>
<thead>
<tr>
<th>NODE/ ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SXX</td>
<td>SYY</td>
</tr>
<tr>
<td>4 1 21</td>
<td>-0.144</td>
<td>-0.449</td>
</tr>
<tr>
<td>4 1 25</td>
<td>-0.132</td>
<td>-0.368</td>
</tr>
<tr>
<td>4 1 26</td>
<td>-0.009</td>
<td>-0.377</td>
</tr>
<tr>
<td>4 1 22</td>
<td>-0.012</td>
<td>-0.449</td>
</tr>
<tr>
<td>4 1 41</td>
<td>-0.152</td>
<td>-0.484</td>
</tr>
<tr>
<td>4 1 45</td>
<td>-0.155</td>
<td>-0.449</td>
</tr>
<tr>
<td>4 1 46</td>
<td>-0.005</td>
<td>-0.449</td>
</tr>
<tr>
<td>4 1 42</td>
<td>0.007</td>
<td>-0.475</td>
</tr>
<tr>
<td>4 1 CENTER</td>
<td>-0.075</td>
<td>-0.437</td>
</tr>
</tbody>
</table>

S1= -0.070  S2= -0.080  S3= -0.438  SE= 0.363

DC= 0.707  S1= 0.707  S2= -0.041  S3= 0.707  SE= 0.707

4 2 21  
0.176  
1.021  
0.284  
0.217  
0.814  
0.005

4 2 25  
0.154  
-0.006  
0.022  
0.251  
0.814  
0.029

4 2 26  
-0.028  
0.053  
-0.015  
0.253  
0.816  
0.002

4 2 22  
-0.054  
1.031  
0.103  
0.219  
0.812  
0.036

4 2 41  
0.189  
1.034  
0.321  
0.258  
0.838  
0.029

4 2 45  
0.162  
-0.006  
0.054  
0.223  
0.010  
0.005

4 2 46  
-0.225  
-0.016  
-0.051  
0.221  
0.008  
0.026

4 2 42  
-0.247  
0.976  
0.071  
0.255  
0.036  
0.060

4 2 CENTER  
0.016  
0.511  
0.099  
0.237  
0.014  
0.015

S1= 0.606  S2= 0.101  S3= -0.082  SE= 0.617

DC= 0.372  S1= 0.372  S2= 0.928  S3= -0.106  SE= 0.994

**EXAMPLE PROBLEM USING SOLID ELEMENTS**  
**-- PAGE NO. 13**

**ELEMENT STRESSES**  
**UNITS= NEWTMMS**

<table>
<thead>
<tr>
<th>NODE/ ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SXX</td>
<td>SYY</td>
</tr>
<tr>
<td>4 3 21</td>
<td>0.900</td>
<td>5.181</td>
</tr>
<tr>
<td>4 3 25</td>
<td>-0.893</td>
<td>-5.740</td>
</tr>
<tr>
<td>4 3 26</td>
<td>5.251</td>
<td>-3.282</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>NODE</th>
<th>ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>3 22</td>
<td>5.379 5.974 1.830 4.029 6.019 1.649</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3 41</td>
<td>2.148 9.507 2.550 0.107 5.891 1.229</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3 45</td>
<td>2.276 4.348 1.292 -1.518 0.596 -0.396</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3 46</td>
<td>-1.334 3.555 2.982 -0.558 -0.364 2.442</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3 42</td>
<td>-3.127 7.049 -0.756 1.067 6.851 0.816</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3 CENTER</td>
<td>1.325 3.324 1.517 2.548 3.244 1.023</td>
<td></td>
</tr>
</tbody>
</table>

- \( S_1 = 7.030 \)
- \( S_2 = 0.411 \)
- \( S_3 = -1.275 \)
- \( S = 7.604 \)

- \( D = 0.425 0.744 0.516 0.809 -0.085 -0.586 \)

<table>
<thead>
<tr>
<th>NODE</th>
<th>ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>3 CENTER</td>
<td>1.325 3.324 1.517 2.548 3.244 1.023</td>
<td></td>
</tr>
</tbody>
</table>

**EXAMPLE PROBLEM USING SOLID ELEMENTS**

<table>
<thead>
<tr>
<th>NODE</th>
<th>ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>3 CENTER</td>
<td>1.325 3.324 1.517 2.548 3.244 1.023</td>
<td></td>
</tr>
</tbody>
</table>

- \( S_1 = 7.030 \)
- \( S_2 = 0.411 \)
- \( S_3 = -1.275 \)
- \( S = 7.604 \)

- \( D = 0.425 0.744 0.516 0.809 -0.085 -0.586 \)

### STAAD.Pro

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>NODE</th>
<th>ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>3 CENTER</td>
<td>1.325 3.324 1.517 2.548 3.244 1.023</td>
<td></td>
</tr>
</tbody>
</table>

- \( S_1 = 7.030 \)
- \( S_2 = 0.411 \)
- \( S_3 = -1.275 \)
- \( S = 7.604 \)

- \( D = 0.425 0.744 0.516 0.809 -0.085 -0.586 \)

**EXAMPLE PROBLEM USING SOLID ELEMENTS**

<table>
<thead>
<tr>
<th>NODE</th>
<th>ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>3 CENTER</td>
<td>1.325 3.324 1.517 2.548 3.244 1.023</td>
<td></td>
</tr>
</tbody>
</table>

- \( S_1 = 7.030 \)
- \( S_2 = 0.411 \)
- \( S_3 = -1.275 \)
- \( S = 7.604 \)

- \( D = 0.425 0.744 0.516 0.809 -0.085 -0.586 \)
### Application Examples

**EX. British Design Examples**

#### ELEMENT LOAD CENTER

<table>
<thead>
<tr>
<th>ELEMENT</th>
<th>LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>1</td>
<td>44</td>
<td>-0.259</td>
<td>-0.089</td>
<td>-0.259</td>
<td>0.273</td>
<td>0.273</td>
</tr>
<tr>
<td>S1=</td>
<td>0.027</td>
<td>S2=</td>
<td>0.044</td>
<td>S3=</td>
<td>-0.368</td>
<td>SE=</td>
<td>0.365</td>
</tr>
<tr>
<td>DC=</td>
<td>0.631</td>
<td>-0.451</td>
<td>0.631</td>
<td>-0.707</td>
<td>-0.000</td>
<td>0.707</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>23</td>
<td>-0.032</td>
<td>0.112</td>
<td>-0.001</td>
<td>0.030</td>
<td>0.002</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>27</td>
<td>-0.001</td>
<td>0.025</td>
<td>-0.046</td>
<td>0.073</td>
<td>0.013</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>28</td>
<td>0.096</td>
<td>0.003</td>
<td>-0.065</td>
<td>0.083</td>
<td>0.003</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>24</td>
<td>-0.085</td>
<td>0.177</td>
<td>0.109</td>
<td>0.040</td>
<td>0.012</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>43</td>
<td>-0.152</td>
<td>0.158</td>
<td>0.052</td>
<td>0.136</td>
<td>0.023</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>47</td>
<td>-0.140</td>
<td>0.041</td>
<td>-0.013</td>
<td>0.092</td>
<td>0.008</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>48</td>
<td>-0.496</td>
<td>-0.041</td>
<td>-0.013</td>
<td>0.092</td>
<td>0.008</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>CENTER</td>
<td>-0.183</td>
<td>0.051</td>
<td>-0.001</td>
<td>0.083</td>
<td>-0.007</td>
</tr>
<tr>
<td>S1=</td>
<td>0.081</td>
<td>S2=</td>
<td>-0.001</td>
<td>S3=</td>
<td>-0.213</td>
<td>SE=</td>
<td>0.263</td>
</tr>
<tr>
<td>DC=</td>
<td>0.314</td>
<td>0.928</td>
<td>-0.202</td>
<td>-0.060</td>
<td>0.232</td>
<td>0.971</td>
<td></td>
</tr>
</tbody>
</table>

#### EXAMPLE PROBLEM USING SOLID ELEMENTS

---

### Application Examples

**EX. British Design Examples**

#### ELEMENT STRESSES

<table>
<thead>
<tr>
<th>ELEMENT LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>4</td>
<td>23</td>
<td>0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>27</td>
<td>0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>28</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>24</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>43</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>47</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>48</td>
<td>-0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
<tr>
<td>6</td>
<td>4</td>
<td>CENTER</td>
<td>0.000</td>
<td>-0.024</td>
<td>0.000</td>
<td>-0.000</td>
</tr>
</tbody>
</table>

---

### Application Examples

**EX. British Design Examples**

#### ELEMENT STRESSES

<table>
<thead>
<tr>
<th>ELEMENT LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>5</td>
<td>23</td>
<td>0.296</td>
<td>-0.507</td>
<td>0.412</td>
<td>-0.013</td>
</tr>
<tr>
<td>6</td>
<td>10</td>
<td>27</td>
<td>-0.071</td>
<td>-1.764</td>
<td>-0.117</td>
<td>-0.025</td>
</tr>
</tbody>
</table>

---

**EX. British Design Examples**

#### ELEMENT STRESSES

<table>
<thead>
<tr>
<th>ELEMENT LOAD CENTER</th>
<th>SXX</th>
<th>SYY</th>
<th>SZZ</th>
<th>SXY</th>
<th>SYZ</th>
<th>SZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>10</td>
<td>23</td>
<td>0.296</td>
<td>0.507</td>
<td>0.412</td>
<td>-0.013</td>
</tr>
<tr>
<td>6</td>
<td>10</td>
<td>27</td>
<td>-0.071</td>
<td>-1.764</td>
<td>-0.117</td>
<td>-0.025</td>
</tr>
</tbody>
</table>
### Example Problem Using Solid Elements

**ELEMENT STRESSES**

**UNITS= NEWTMMS**

<table>
<thead>
<tr>
<th>NODE/ ELEMENT LOAD CENTER</th>
<th>NORMAL STRESSES</th>
<th>SHEAR STRESSES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SXX</td>
<td>SYY</td>
</tr>
<tr>
<td>6 10 28</td>
<td>-0.773</td>
<td>-1.853</td>
</tr>
<tr>
<td>6 10 24</td>
<td>-0.251</td>
<td>0.572</td>
</tr>
<tr>
<td>6 10 43</td>
<td>-0.249</td>
<td>-0.043</td>
</tr>
<tr>
<td>6 10 47</td>
<td>0.272</td>
<td>0.354</td>
</tr>
<tr>
<td>6 10 48</td>
<td>-0.356</td>
<td>0.289</td>
</tr>
<tr>
<td>6 10 44</td>
<td>-0.724</td>
<td>0.047</td>
</tr>
<tr>
<td>6 10 CENTER</td>
<td>-0.232</td>
<td>-0.236</td>
</tr>
</tbody>
</table>

Total:

- S1 = 0.011
- S2 = -0.212
- S3 = -0.316
- SE = 0.289

DC = -0.080, -0.428, 0.900, 0.888, -0.441, -0.131

**62. FINISH**

*********** END OF THE STAAD.Pro RUN ***********

**** DATE= APR 14,2019    TIME= 22:53:58 ****

Example Problem Using Solid Elements

---

**Related Links**

- *M. To assign an enforced support* (on page 813)
- *TR.27.1 Global Support Specification* (on page 2514)
- *Create Support dialog* (on page 2983)

---

**EX. UK-25 Analysis of a Structure with Compression-Only Members**

This example demonstrates the usage of compression-only members. Since the structural condition is load dependent, the **PERFORM ANALYSIS** command is specified once for each primary load case.

This problem is installed with the program by default to

C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-25 Analysis of a Structure with Compression-Only Members.STD when you install the program.
This example has been created to illustrate the command specification for a structure with certain members capable of carrying compressive force only. It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of “active” members (and hence the stiffness matrix) is load case dependent.

where:

\[
L = 5.25 \text{ m}, \quad H = 3.5 \text{ m} \\
\text{Load case 1: } P_1 = 45 \text{ kN} \quad \text{&} \quad P_2 = 70 \text{ kN} \\
\text{Load case 2: } P_3 = -45 \text{ kN} \quad \text{&} \quad P_4 = -70 \text{ kN}
\]

STAAD PLANE

* EXAMPLE FOR COMPRESSION-ONLY MEMBERS

The input data is initiated with the word STAAD. This structure is a PLANE frame. The second line is an optional comment line.

UNIT METER KNS

Units for the commands to follow are specified above.

JOINT COORDINATES
1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
Joint coordinates of joints 1 to 6 are defined above.

```
MEMBER INCIDENCES
  1 1 2 5 ; 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

The members 1 to 10 are defined along with the joints to which they are connected.

```
MEMBER COMPRESSION
  6 TO 9
```

Members 6 to 9 are defined as COMPRESSION-only members. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a tensile force, it is disabled from the structure and the analysis is carried out again with the modified structure.

```
MEMBER PROPERTY BRITISH
  1 TO 10 TA ST UC152X152X30
```

Members 1 to 10 are assigned the UC152X152X30 section from the British steel table.

```
UNIT MMS
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 210
  POISSON 0.3
  DENSITY 7.6977e-008
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
  MATERIAL STEEL ALL
  UNIT METER
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
SUPPORT
  1 6 PINNED
```

Joints 1 and 6 are declared as pinned-supported.

```
LOAD 1
  JOINT LOAD
  2 FX 70
  3 FX 45
```

Load 1 is defined above and consists of joint loads in the global X direction at joints 2 and 3.

```
PERFORM ANALYSIS
```

The above structure is analyzed for load case 1.

```
CHANGE
  MEMBER COMPRESSION
  6 TO 9
```

One or more among the members 6 to 9 may have been in-activated in the previous analysis. The CHANGE command restores the original structure to prepare it for the analysis for the next primary load case. The members with the compression-only attribute are specified again.

```
LOAD 2
  JOINT LOAD
```
In load case 2, joint loads are applied in the negative global X direction at joints 4 and 5.

```
PERFORM ANALYSIS
CHANGE
```

The instruction to analyze the structure is specified again. Next, any compression-only members that were inactivated during the second analysis (due to the fact that they were subjected to tensile axial forces) are re-activated with the CHANGE command. Without the re-activation, these members cannot be accessed for further processing.

```
LOAD 3
REPEAT LOAD
1 1.0 2 1.0
```

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously. In other words, the above instruction is the same as the following:

```
LOAD 3
JOINT LOAD
2 FX 70
3 FX 45
4 FX -45
5 FX -70
PERFORM ANALYSIS
```

The analysis is carried out for load case 3.

```
CHANGE
```

The members inactivated during the analysis of load case 3 are re-activated for further processing.

```
LOAD LIST ALL
```

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the “active” load cases. The LOAD LIST ALL command enables all the load cases in the structure to be made active for further processing.

```
PRINT ANALYSIS RESULTS
```

The program is instructed to write the joint displacements, support reactions and member forces to the output file.

```
FINISH
```

The STAAD run is terminated.

### Input File

```
STAAD PLANE
* EXAMPLE FOR COMPRESSION-ONLY MEMBERS
UNIT METER KNS
SET NL 3
JOINT COORDINATES
1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

MEMBER COMPRESSION
6 TO 9
MEMBER PROPERTY BRITISH
1 TO 10 TA ST UC152X152X30
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
SUPPORT
1 6 PINNED
LOAD 1
JOINT LOAD
2 FX 70
3 FX 45
PERFORM ANALYSIS
CHANGE
MEMBER COMPRESSION
6 TO 9
LOAD 2
JOINT LOAD
4 FX -45
5 FX -70
PERFORM ANALYSIS
CHANGE
LOAD 3
REPEAT LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ANALYSIS RESULTS
FINISH

STAAD Output File
INPUT FILE: UK-25 Analysis of a Structure with Compression-Only Members.STD

2. * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
3. UNIT METER KNS
4. SET NL 3
5. JOINT COORDINATES
   6. 1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
7. MEMBER INCIDENCES
8. 1 1 2 5
9. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
10. MEMBER COMPRESSION
11. 6 TO 9
12. MEMBER PROPERTY BRITISH
13. 1 TO 10 TA ST UC152X152X30
14. UNIT MMS
15. DEFINE MATERIAL START
16. ISOTROPIC STEEL
17. E 210
18. POISSON 0.3
19. DENSITY 7.6977E-008
20. ALPHA 6E-006
21. DAMP 0.03
22. TYPE STEEL
23. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
24. END DEFINE MATERIAL
25. CONSTANTS
26. MATERIAL STEEL ALL
27. UNIT METER
28. SUPPORT
29. 1 6 PINNED
30. LOAD 1
31. JOINT LOAD
32. 2 FX 70
33. 3 FX 45
34. PERFORM ANALYSIS

* EXAMPLE FOR COMPRESSION-ONLY MEMBERS

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS          6  NUMBER OF MEMBERS      10
NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      10
Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
*** LOAD CASE     1 -- START ITERATION NO.   2
**NOTE-Tension/Compression converged after  2 iterations, Case=    1
35. CHANGE
36. MEMBER COMPRESSION
37. 6 TO 9
38. LOAD 2
39. JOINT LOAD
40. 4 FX -45
41. 5 FX -70
42. PERFORM ANALYSIS
*** LOAD CASE     2 -- START ITERATION NO.   2
**NOTE-Tension/Compression converged after  2 iterations, Case=    2
43. CHANGE
**NOTE-Tension/Compression converged after 1 iterations, Case= 3**

### Joint Displacement (cm radians)

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-Rotan</th>
<th>Y-Rotan</th>
<th>Z-Rotan</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0005</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>2</td>
<td>0.1609</td>
<td>0.0464</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>3</td>
<td>-0.1900</td>
<td>-0.0132</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0138</td>
<td>0.0030</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
</tr>
<tr>
<td>4</td>
<td>0.2889</td>
<td>0.0594</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0003</td>
</tr>
<tr>
<td>3</td>
<td>-0.2889</td>
<td>-0.0132</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0002</td>
</tr>
<tr>
<td>3</td>
<td>0.0991</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
</tr>
</tbody>
</table>

### Member End Forces

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-106.67</td>
<td>0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>106.67</td>
<td>-0.13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.45</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>30.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-30.22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2.94</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>6.94</td>
<td>-0.03</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>29.91</td>
<td>0.24</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.38</td>
<td></td>
</tr>
<tr>
<td>MEMBER</td>
<td>LOAD</td>
<td>JT</td>
<td>AXIAL</td>
<td>SHEAR-Y</td>
<td>SHEAR-Z</td>
<td>TORSION</td>
<td>MOM-Y</td>
<td>MOM-Z</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td>----</td>
<td>-------</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
<td>-------</td>
<td>-------</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>12.51</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>137.81</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>-137.81</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>-20.72</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>-12.51</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>12.51</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>137.81</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>-137.81</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>53.66</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>-53.66</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>42.29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

* Application Examples

EX. British Design Examples

**MEMBER END FORCES**

**STRUCTURE TYPE = PLANE**

---

**ALL UNITS ARE -- KNS METE (LOCAL )**

---

**STAAD PLANE**

---

**-- PAGE NO. 7**

*EXAMPLE FOR COMPRESSION-ONLY MEMBERS*
EX. UK-26 Modeling a Rigid Diaphragm Using Master-Slave

The structure in this example is a building consisting of member columns as well as floors made up of beam members and plate elements. Using the master-slave command, the floors are specified to be rigid diaphragms for in-plane actions but flexible for bending actions.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-26 Modeling a Rigid Diaphragm Using Master-Slave.STD when you install the program.
Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The second line is an optional title to identify this project.

Specify units for the following data.

| JOINT COORD | 1 0 0 0 4 0 48 0 |
| REPEAT 3 24 0 0 |
| REPEAT ALL 3 0 0 24 |
| DELETE JOINT 21 25 37 41 |

The joint numbers and coordinates are specified above. The unwanted joints, created during the generation process used above, are then deleted.

| MEMBER INCI | 1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12 |
| 13 17 18 15 ; 22 29 30 24 ; 25 33 34 27 |
| 34 45 46 36 ; 37 49 50 39 ; 40 53 54 42 |
| 43 57 58 45 ; 46 61 62 48 ; 49 2 6 51 |
| 52 6 10 54 ; 55 10 14 57 ; 58 18 22 60 |
| 61 22 26 63 ; 64 26 30 66 ; 67 34 38 69 |
The **MEMBER INCIDENCE** specification is used for specifying MEMBER connectivities.

```
    ELEMENT INCI
    152 50 34 38 54 TO 154
    155 54 38 42 58 TO 157
    158 58 42 46 62 TO 160
    161 34 18 22 38 TO 163
    164 38 22 26 42 TO 166
    167 42 26 30 46 TO 169
    170 18 2 6 22 TO 172
    173 22 6 10 26 TO 175
    176 26 10 14 30 TO 178
```

The **ELEMENT INCIDENCE** specification is used for specifying plate element connectivities.

```
    MEMBER PROPERTIES AMERICAN
    1 TO 15 22 TO 27 34 TO 48 TABLE ST W14X90
    49 TO 120 TABLE ST W27X84
```

All members are **WIDE FLANGE** sections whose properties are obtained from the built in American steel table.

```
    ELEMENT PROP
    152 TO 178 THICK 0.75
```

The thickness of the plate elements is specified above.

```
    UNIT INCHES
    DEFINE MATERIAL START
    ISOTROPIC STEEL
    E 29000
    POISSON 0.3
    DENSITY 283e-006
    ALPHA 6e-006
    DAMP 0.03
    TYPE STEEL
    STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    ISOTROPIC CONCRETE
    E 3150
    POISSON 0.17
    DENSITY 8.68e-005
    ALPHA 5e-006
    DAMP 0.05
    G 1346.15
    TYPE CONCRETE
    STRENGTH FCU 4
    END DEFINE MATERIAL
    UNIT FEET
    MATERIAL STEEL MEMB 1 TO 15 22 TO 27 34 TO 120
    BETA 90.0 MEMB 13 14 15 22 TO 27 34 TO 39
    MATERIAL CONCRETE MEMB 152 TO 178
```
The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. The orientation of some of the members is set using the BETA angle command.

<table>
<thead>
<tr>
<th>SUPPORTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 TO 17 BY 4 29 33 45 TO 61 BY 4 FIXED</td>
</tr>
</tbody>
</table>

The supports at the above mentioned joints are declared as fixed.

| SLAVE DIA ZX MASTER 22 JOINTS YR 15.0 17.0 |
| SLAVE DIA ZX MASTER 23 JOINTS YR 31.0 33.0 |
| SLAVE DIA ZX MASTER 24 JOINTS YR 47.0 49.0 |

The three floors of the structure are specified to act as rigid diaphragms in the ZX plane with the corresponding master joint specified. The associated slave joints in a floor are specified by the YRANGE parameter. The floors may still resist out-of-plane bending actions flexibly.

<table>
<thead>
<tr>
<th>LOADING 1 LATERAL LOADS</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOADS</td>
</tr>
<tr>
<td>2 3 4 14 15 16 50 51 52 62 63 64 FZ 10.0</td>
</tr>
<tr>
<td>6 7 8 10 11 12 18 19 20 30 31 32 FZ 20.0</td>
</tr>
<tr>
<td>34 35 36 46 47 48 54 55 56 58 59 60 FZ 20.0</td>
</tr>
<tr>
<td>22 23 24 26 27 28 38 39 40 42 43 44 FZ 40.0</td>
</tr>
</tbody>
</table>

The above data describe a static load case. It consists of joint loads in the global Z direction.

<table>
<thead>
<tr>
<th>LOADING 2 TORSIONAL LOADS</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOINT LOADS</td>
</tr>
<tr>
<td>2 3 4 50 51 52 FZ 5.0</td>
</tr>
<tr>
<td>14 15 16 62 63 64 FZ 15.0</td>
</tr>
<tr>
<td>6 7 8 18 19 20 FZ 10.0</td>
</tr>
<tr>
<td>10 11 12 30 31 32 FZ 30.0</td>
</tr>
<tr>
<td>34 35 36 54 55 56 FZ 10.0</td>
</tr>
<tr>
<td>46 47 48 58 59 60 FZ 30.0</td>
</tr>
<tr>
<td>22 23 24 38 39 40 FZ 20.0</td>
</tr>
<tr>
<td>26 27 28 42 43 44 FZ 60.0</td>
</tr>
</tbody>
</table>

The above data describe a static load case. It consists of joint loads that create a torsional loading on the structure.

<table>
<thead>
<tr>
<th>LOADING 3 DEAD LOAD</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEMENT LOAD</td>
</tr>
<tr>
<td>152 TO 178 PRESS GY -1.0</td>
</tr>
</tbody>
</table>

The above data describe a static load case. It consists of plate element pressure on a floor in the negative global Y direction.

| PERFORM ANALYSIS |

The above command instructs the program to proceed with the analysis.

| PRINT JOINT DISP LIST 4 TO 60 BY 8 |
| PRINT MEMBER FORCES LIST 116 115 |
| PRINT SUPPORT REACTIONS LIST 9 57 |

Print displacements at selected joints, then print member forces for two members, then print support reactions at selected joints.

| FINISH |

The STAAD run is terminated.
Input File

STAAD SPACE
*MODELING RIGID DIAPHRAGMS USING MASTER SLAVE
UNITS KIP FT
JOINT COORD
1 0 0 4 0 48 0
REPEAT 3 24 0 0
REPEAT ALL 3 0 0 24
DELETE JOINT 21 25 37 41

MEMBER INCI
1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12
13 17 18 15 ; 22 29 30 24 ; 25 33 34 27
34 45 46 36 ; 37 49 50 39 ; 40 53 54 42
43 57 58 45 ; 46 61 62 48 ; 49 2 6 51
52 6 10 54 ; 55 10 14 57 ; 58 18 22 60
61 22 26 63 ; 64 26 30 66 ; 67 34 38 69
70 38 42 72 ; 73 42 46 75 ; 76 50 54 78
79 54 58 81 ; 82 58 62 84 ; 85 18 2 87
88 22 6 90 ; 91 26 10 93 ; 94 30 14 96
97 34 18 99 ; 100 38 22 102 ; 103 42 26 105
106 46 30 108 ; 109 50 34 111 ; 112 54 38 114
115 58 42 117 ; 118 62 46 120

ELEMENT INCI
152 50 34 38 54 TO 154
155 54 38 42 58 TO 157
158 58 42 46 62 TO 160
161 34 18 22 38 TO 163
164 38 22 26 42 TO 166
167 42 26 30 46 TO 169
170 18 2 6 22 TO 172
173 22 6 10 26 TO 175
176 26 10 14 30 TO 178

MEMBER PROPERTIES AMERICAN
1 TO 15 22 TO 27 34 TO 48 TA ST W14X90
49 TO 120 TABLE ST W27X84

ELEMENT PROP
152 TO 178 THICK 0.75
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2

ISOTROPIC CONCRETE
E 3150
E 2916.7
POISSON 0.17
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 1 TO 15 22 TO 27 34 TO 120
BETA 90.0 MEMB 13 14 15 22 TO 27 34 TO 39
MATERIAL CONCRETE MEMB 152 TO 178
SUPPORTS
1 TO 17 BY 4 29 33 45 TO 61 BY 4 FIXED
SLAVE DIA ZX MASTER 22 JOINTS YR 15.0 17.0
SLAVE DIA ZX MASTER 23 JOINTS YR 31.0 33.0
SLAVE DIA ZX MASTER 24 JOINTS YR 47.0 49.0
LOADING 1 LATERAL LOADS
JOINT LOADS
2 3 4 14 15 16 50 51 52 62 63 64 FZ 10.0
6 7 8 10 11 12 18 19 20 30 31 32 FZ 20.0
34 35 36 46 47 48 54 55 56 58 59 60 FZ 20.0
22 23 24 26 27 28 38 39 40 42 43 44 FZ 40.0
LOADING 2 TORSIONAL LOADS
JOINT LOADS
2 3 4 50 51 52 FZ 5.0
14 15 16 62 63 64 FZ 15.0
6 7 8 18 19 20 FZ 10.0
10 11 12 30 31 32 FZ 30.0
34 35 36 54 55 56 FZ 10.0
46 47 48 58 59 60 FZ 30.0
22 23 24 38 39 40 FZ 20.0
26 27 28 42 43 44 FZ 60.0
LOADING 3 DEAD LOAD
ELEMENT LOAD
152 TO 178 PRESS GY -1.0
PERFORM ANALYSIS
PRINT JOINT DISP LIST 4 TO 60 BY 8
PRINT MEMBER FORCES LIST 116 115
PRINT SUPPORT REACTIONS LIST 9 57
FINISH

STAAD Output File
7. REPEAT ALL 3 0 0 24
8. DELETE JOINT 21 25 37 41
9. MEMBER INCI
10. 1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12
11. 13 17 18 15 ; 22 29 30 24 ; 25 33 34 27
12. 34 45 46 36 ; 37 49 50 39 ; 40 53 54 42
13. 43 57 58 45 ; 46 61 62 48 ; 49 2 6 51
14. 52 6 10 54 ; 55 10 14 57 ; 58 18 22 60
15. 61 22 26 63 ; 64 26 30 66 ; 67 34 38 69
16. 70 38 42 72 ; 73 42 46 75 ; 76 50 54 78
17. 79 54 58 81 ; 82 58 62 84 ; 85 18 2 87
18. 88 22 6 90 ; 91 26 10 93 ; 94 30 14 96
19. 97 34 18 99 ; 100 38 22 102 ; 103 42 26 105
20. 106 46 30 108 ; 109 50 34 111 ; 112 54 38 114
21. 115 58 42 117 ; 118 62 46 120
22. ELEMENT INCI
23. 152 50 34 38 54 TO 154
24. 155 54 38 42 58 TO 157
25. 158 58 42 46 62 TO 160
26. 161 34 18 22 38 TO 163
27. 164 38 22 26 42 TO 166
28. 167 42 26 30 46 TO 169
29. 170 18 2 6 22 TO 172
30. 173 22 6 10 26 TO 175
31. 176 26 10 14 30 TO 178
32. MEMBER PROPERTIES AMERICAN
33. 1 TO 15 22 TO 27 34 TO 48 TA ST W14X90
34. 49 TO 120 TABLE ST W27X84
35. ELEMENT PROP
36. 152 TO 178 THICK 0.75
37. UNIT INCHES
38. DEFINE MATERIAL START
  STAAD SPACE
*MODELING RIGID DIAPHRAGMS USING MASTER SLAV
  39. ISOTROPIC STEEL
  40. E 29000
  41. POISSON 0.3
  42. DENSITY 283E-006
  43. ALPHA 6E-006
  44. DAMP 0.03
  45. TYPE STEEL
  46. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
  47. ISOTROPIC CONCRETE
  48. E 3150
  49. E 2916.7
  50. POISSON 0.17
  51. POISSON 0.12
  52. DENSITY 8.68E-005
  53. ALPHA 5E-006
  54. DAMP 0.05
  55. G 1346.15
  56. TYPE CONCRETE
  57. STRENGTH FCU 4
  58. END DEFINE MATERIAL
  59. CONSTANTS
  60. MATERIAL STEEL MEMB 1 TO 15 22 TO 27 34 TO 120
  61. BETA 90.0 MEMB 13 14 15 22 TO 27 34 TO 39
  62. MATERIAL CONCRETE MEMB 152 TO 178
Related Links

- M. To display master nodes (on page 638)

**EX. UK-27 Modeling Soil Springs for a Slab on Grade**

This example illustrates the usage of commands necessary to apply the compression only attribute to spring supports for a slab on grade. The spring supports themselves are generated utilizing the built-in support generation facility. The slab is subjected to pressure and overturning loading. A tension/compression only analysis of the structure is performed.

This problem is installed with the program by default to 
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-27 Modeling Soil Springs for a Slab on Grade.STD when you install the program.

The numbers shown in the diagram below are the element numbers.
where:

\[
H = 129', h_1 = 8'-6'', h_2 = 8', h_3 = 6'
\]
\[
V = 40', v_1 = 6'-6'', v_2 = 6', v_3 = 7'-10'', v_4 = 4'
\]

STAAD SPACE SLAB ON GRADE
* SPRING COMPRESSION EXAMPLE

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. An optional title to identify this project is provided in the second line.

SET NL 3

This structure has to be analyzed for 3 primary load cases. Consequently, the modeling of our problem requires us to define 3 sets of data, with each set containing a load case and an associated analysis command. Also, the supports which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have 2 commands: SET NL and CHANGE. The SET NL command is used above to indicate the total number of primary load cases that the file contains. The CHANGE command will come in later (after the PERFORM ANALYSIS command).
For joints 1 through 7, the joint number followed by the X, Y and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these 7 joints by 8.5 feet, 3 times. REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

```
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
```

The incidences of element number 1 is defined and this data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first 6 elements is then used to generate the incidences of 96 more elements using the REPEAT command.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 8.0
```

The thickness of elements 1 to 102 is specified as 8.0 inches following the command ELEMENT PROPERTIES.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 4000.0
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The modulus of elasticity (E) and Poisson's Ratio are specified following the command CONSTANTS.

```
SPRING COMPRESSION
1 TO 126 KFY
```

The above two lines declare the spring supports at nodes 1 to 126 as having the compression-only attribute. The supports themselves are being generated later (see the ELASTIC MAT command which appears later).

```
UNIT FEET
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 12.0
```

The above command is used to instruct STAAD to generate supports with compression-only springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade reaction of the soil is specified as 12 kip/cu.ft. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the "FY" degree of freedom at the joint. Units for length are changed to FEET to facilitate the input of subgrade reaction of soil. See TR.27.3 Automatic Spring Support Generator for Foundations (on page 2517).

```
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.50
```

The above data describe a static load case. A pressure load of 1.50 kip/ft acting in the negative global Y direction is applied on all the elements.

```
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
```
Tension/compression cases must each be followed by `PERFORM ANALYSIS` and `CHANGE` commands. The `CHANGE` command restores the original structure to prepare it for the analysis for the next primary load case.

```
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
  1 2 FY -217.
  8 9 FY -109.
  5 FY -308.7
  6 FY -617.4
 22 23 FY -410.
 29 30 FY -205.
 26 FY -542.7
 27 FY -1085.4
 43 44 50 51 71 72 78 79 FY -307.5
 47 54 82 FY -264.2
 48 55 76 83 FY -528.3
 92 93 FY -205.0
 99 100 FY -410.0
 103 FY -487.0
 104 FY -974.0
 113 114 FY -109.0
 120 121 FY -217.0
 124 FY -273.3
FY -546.6
PERFORM ANALYSIS PRINT STATICS CHECK
```

Load case 2 consists of several joint loads acting in the negative global Y direction. This is followed by another `ANALYSIS` command. The `CHANGE` command restores the original structure once again for the forthcoming load case.

```
LOAD 3 'COLUMN OVERTURNING LOAD'
ELEMENT LOAD
  1 TO 102 PR GY -1.50
JOINT LOADS
  1 2 FY -100.
  8 9 FY -50.
  5 FY -150.7
  6 FY -310.4
 22 23 FY -205.
 29 30 FY -102.
 26 FY -271.7
 27 FY -542.4
 43 44 50 51 71 72 78 79 FY -153.5
 47 54 82 FY -132.2
 48 55 76 83 FY -264.3
 92 93 FY 102.0
 99 100 FY 205.0
103 FY 243.0
 104 FY 487.0
 113 114 FY 54.0
120 121 FY 108.0
124 FY 136.3
125 FY 273.6
PERFORM ANALYSIS PRINT STATICS CHECK
```
Load case 3 consists of several joint loads acting in the upward direction at one end and downward on the other end to apply an overturning moment that will lift off one end. The CHANGE command is not needed after the last analysis.

LOAD LIST 3
PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
PRINT ELEMENT STRESSES LIST 34 67
PRINT SUPPORT REACTIONS LIST 5 6 12 13

A list of joint displacements, element stresses for elements 34 and 67, and support reactions at a list of joints for load case 3, are obtained with the help of the above commands.

FINISH

The STAAD run is terminated.

Input File

STAAD SPACE SLAB ON GRADE
* SPRING COMPRESSION EXAMPLE
SET NL 3
UNIT FEET KIP
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0 6.5
7 0.0 0.0 0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 8.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 4000.0
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SPRING COMPRESSION
1 TO 126 KFY
UNIT FEET
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 12.0
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.50
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2 FY -217.
8 9 FY -109.
5 FY -308.7
6 FY -617.4
22 23 FY -410.
29 30 FY -205.
26 FY -542.7
27 FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82 FY -264.2
48 55 76 83 FY -528.3
92 93 FY -205.0
99 100 FY -410.0
103 FY -487.0
104 FY -974.0
113 114 FY -109.0
120 121 FY -217.0
124 FY -273.3
125 FY -546.6
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 3 'COLUMN OVERTURNING LOAD'
ELEMENT LOAD
1 TO 102 PR GY -1.50
JOINT LOADS
1 2 FY -100.
8 9 FY -50.
5 FY -150.7
6 FY -310.4
22 23 FY -205.
29 30 FY -102.
26 FY -271.7
27 FY -542.4
43 44 50 51 71 72 78 79 FY -153.5
47 54 82 FY -132.2
48 55 76 83 FY -264.3
92 93 FY 102.0
99 100 FY 205.0
103 FY 243.0
104 FY 487.0
113 114 FY 54.0
120 121 FY 108.0
124 FY 136.3
125 FY 273.6
PERFORM ANALYSIS PRINT STATICS CHECK
LOAD LIST 3
PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
PRINT ELEMENT STRESSES LIST 34 67
PRINT SUPPORT REACTIONS LIST 5 6 12 13
FINISH
1. STAAD SPACE SLAB ON GRADE

INPUT FILE: UK-27 Modeling Soil Springs for a Slab on Grade.STD

2. * SPRING COMPRESSION EXAMPLE

3. SET NL 3
4. UNIT FEET KIP
5. JOINT COORDINATES
6. 1 0.0 0.0 40.0
7. 2 0.0 0.0 36.0
8. 3 0.0 0.0 28.167
9. 4 0.0 0.0 20.333
10. 5 0.0 0.0 12.5
11. 6 0.0 0.0  6.5
12. 7 0.0 0.0  0.0
13. REPEAT ALL 3 8.5 0.0 0.0
14. REPEAT 3 8.0 0.0 0.0
15. REPEAT 5 6.0 0.0 0.0
16. REPEAT 3 8.0 0.0 0.0
17. REPEAT 3 8.5 0.0 0.0
18. ELEMENT INCIDENCES
19. 1 1 8 9 2 TO 6
20. REPEAT 16 6 7
21. UNIT INCH
22. ELEMENT PROPERTIES
23. 1 TO 102 TH 8.0
24. DEFINE MATERIAL START
25. ISOTROPIC CONCRETE
26. E 4000.0
27. POISSON 0.12
28. DENSITY 8.68E-005
29. ALPHA 5E-006
30. DAMP 0.05
31. G 1346.15
32. TYPE CONCRETE
33. STRENGTH FCU 4
34. END DEFINE MATERIAL
35. CONSTANTS
36. MATERIAL CONCRETE ALL
37. SPRING COMPRESSION
38. 1 TO 126 KFY

SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE

39. UNIT FEET
40. SUPPORTS
41. 1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 12.0
42. LOAD 1 'WEIGHT OF MAT & EARTH'
43. ELEMENT LOAD
44. 1 TO 102 PR GY -1.50
45. PERFORM ANALYSIS PRINT STATICS CHECK

PROBLEM STATISTICS
-----------------------------------
NUMBER OF JOINTS        126  NUMBER OF MEMBERS       0
NUMBER OF PLATES        102  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS    126

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     378
TOTAL LOAD COMBINATION CASES =     0  SO FAR.

SLAB ON GRADE -- PAGE NO. 3
* SPRING COMPRESSION EXAMPLE
**NOTE-Tension/Compression converged after 1 iterations, Case=    1
STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.  1
'WEIGHT OF MAT & EARTH'
CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X =  0.645000019E+02
Y =  0.000000000E+00
Z =  0.200000006E+02

TOTAL APPLIED LOAD     1
***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING  1)
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =       -7740.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=      154800.01  MY=           0.00  MZ=     -499230.03

TOTAL REACTION LOAD    1
***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING  1)
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =        7740.00
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=     -154800.01  MY=           0.00  MZ=      499230.03

MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING  1)
MAXIMUMS    AT NODE
X =  0.00000E+00       0
Y = -1.50000E+00       1
Z =  0.00000E+00       0
RX=  9.79653E-10     121
RY=  0.00000E+00      45
RZ=  4.32018E-10

************ END OF DATA FROM INTERNAL STORAGE ************

46. CHANGE
47. LOAD 2 'COLUMN LOAD-DL+LL'
48. JOINT LOADS
49. 1 2   FY -217.
50. 8 9   FY -109.
51. 5     FY -308.7
52. 6     FY -617.4
53. 22 23 FY -410.
54. 29 30 FY -205.

SLAB ON GRADE -- PAGE NO. 4
* SPRING COMPRESSION EXAMPLE
PERFORM ANALYSIS PRINT STATICS CHECK

SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE
**NOTE-Tension/Compression converged after 1 iterations, Case=    2

STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.  2
'COLUMN LOAD-DL+IL'
CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)

X =  0.633725605E+02
Y =  0.000000000E+00
Z =  0.215722031E+02

TOTAL APPLIED LOAD  2
***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING 2 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =      -13964.90
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=      301253.66  MY=           0.00  MZ=     -884991.47

TOTAL REACTION LOAD  2
***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING 2 )
SUMMATION FORCE-X =           0.00
SUMMATION FORCE-Y =       13964.90
SUMMATION FORCE-Z =           0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=     -301253.66  MY=           0.00   MZ=      884991.47

MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING 2)
MAXIMUMS    AT NODE
X =  0.00000E+00       0
Y = -1.12956E+01     120
Z =  0.00000E+00       0
RX=  8.11932E-02      99
RY=  0.00000E+00       0
RZ=  1.00569E-01       6

************ END OF DATA FROM INTERNAL STORAGE ************

LOAD 3 'COLUMN OVERTURNING LOAD'

SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE
Application Examples
EX. British Design Examples

78. 22 23 FY -205.
79. 29 30 FY -102.
80. 26 FY -271.7
81. 27 FY -542.4
82. 43 44 50 51 71 72 78 79 FY -153.5
83. 47 54 82 FY -264.3
84. 92 93 FY 102.0
85. 99 100 FY 205.0
86. 103 FY 243.0
87. 104 FY 487.0
88. 113 114 FY 54.0
89. 120 121 FY 108.0
90. 124 FY 136.3
91. 125 FY 273.6
92. PERFORM ANALYSIS PRINT STATICS CHECK
SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE

*** LOAD CASE 3 -- START ITERATION NO. 2
*** LOAD CASE 3 -- START ITERATION NO. 3
*** LOAD CASE 3 -- START ITERATION NO. 4
*** LOAD CASE 3 -- START ITERATION NO. 5
**NOTE-Tension/Compression converged after 5 iterations, Case=  3
STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.  3
`COLUMN OVERTURNING LOAD
CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
(FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
X = 0.45460478E+02
Y = 0.000000000E+00
Z = 0.202712741E+02
TOTAL APPLIED LOAD

***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING 3)
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = -10533.10
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= 213519.36 MY= 0.00 MZ= -478687.78
TOTAL REACTION LOAD

***TOTAL REACTION LOAD( KIP FEET ) SUMMARY (LOADING 3)
SUMMATION FORCE-X = 0.00
SUMMATION FORCE-Y = 10533.10
SUMMATION FORCE-Z = 0.00
SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX= -213519.36 MY= 0.00 MZ= 478687.78
MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING 3)
MAXIMUMS AT NODE
X = 0.00000E+00 0
Y = 3.06386E+01 120
Z = 0.00000E+00 0
RX= -1.35247E-01 120
RY= 0.00000E+00 0
RZ= 1.12342E-01 125
************ END OF DATA FROM INTERNAL STORAGE ************
JOINT DISPLACEMENT (INCH RADIANS)  STRUCTURE TYPE = SPACE

------------------

JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
113 3   0.00000  20.81368   0.00000  -0.10277   0.00000   0.07382
114 3   0.00000  15.82565   0.00000  -0.10134   0.00000   0.06799
120 3   0.00000  30.63862   0.00000  -0.13525   0.00000   0.10711
121 3   0.00000  24.13106   0.00000  -0.12907   0.00000   0.09243

****************** END OF LATEST ANALYSIS RESULT ******************

96. PRINT ELEMENT STRESSES LIST 34 67

ELEMENT STRESSES LIST 34
SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE

ELEMENT STRESSES  FORCE,LENGTH UNITS= KIP  FEET

-------------------

STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

ELEMENT  LOAD  SQX  SQY  MX  MY  MXY  VONT  VONB  SX  SY  SXY  TRESPAT  TRESPAB
34 3  -4.66  -6.79  2.34  7.95  5.74  164.71  164.71  0.00  0.00  0.00  172.44  172.44

TOP : SMAX= 155.71 SMIN= -16.73 TMAX= 86.22 ANGLE= 58.0
BOTT: SMAX= 16.73 SMIN= -155.71 TMAX= 86.22 ANGLE=-32.0

67 3  32.82  6.55  -63.47  5.72  36.48  1238.45  1238.45  0.00  0.00  0.00  1357.37  1357.37

TOP : SMAX= 288.91 SMIN= -1068.46 TMAX= 678.69 ANGLE= 66.7
BOTT: SMAX= 1068.46 SMIN= -288.91 TMAX= 678.69 ANGLE=-23.3

**** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****

MAXIMUM  MINIMUM  MAXIMUM  MAXIMUM  MAXIMUM
PRINCIPAL  PRINCIPAL  SHEAR  VONMISES  TRESCA
STRESS  STRESS  STRESS  STRESS  STRESS
1.068459E+03 -1.068459E+03  6.786865E+02  1.238454E+03  1.357373E+03

PLATE NO.  67  67  67  67  67
CASE NO.   3  3  3  3  3

******************* END OF ELEMENT FORCES *******************

97. PRINT SUPPORT REACTIONS LIST 5 6 12 13

SUPPORT REACTION LIST 5
SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE

SUPPORT REACTIONS -UNIT KIP  FEET  STRUCTURE TYPE = SPACE

-------------------

JOINT  LOAD  FORCE-X  FORCE-Y  FORCE-Z  MOM-X  MOM-Y  MOM-Z
5 3  0.00  149.37  0.00  0.00  0.00
6 3  0.00  170.57  0.00  0.00  0.00
12 3  0.00  148.42  0.00  0.00  0.00
13 3  0.00  152.40  0.00  0.00  0.00

******************* END OF LATEST ANALYSIS RESULT *******************

98. FINISH

SLAB ON GRADE

* SPRING COMPRESSION EXAMPLE

******** END OF THE STAAD.Pro RUN ********

*** DATE= APR 14,2019  TIME= 22:54:12 ***

*************************************************************************
* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/ *
* Details about additional assistance from *

STAAD.Pro  4872  User Manual
EX. UK-28 Calculation of Modes and Frequencies of a Bridge

This example demonstrates the input required for obtaining the modes and frequencies of the skewed bridge shown in the figure below. The structure consists of piers, pier-cap girders and a deck slab.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-28 Calculation of Modes and Frequencies of a Bridge.std when you install the program.

Figure 511: Example Problem No. 28

STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words forms a title to identify this project.

IGNORE LIST

Further below in this file, we will call element lists in which some element numbers may not actually be present in the structure. We do so because it minimizes the effort involved in fetching the desired elements and reduces the size of the respective commands. To prevent the program from treating that condition (referring to elements which do not exist) as an error, the above command is required.

UNIT METER KN
Application Examples

EX. British Design Examples

The units for the data that follows are specified above.

<table>
<thead>
<tr>
<th>JOINT COORDINATES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;</td>
</tr>
<tr>
<td>11 -1 10 25.165 0 10 0</td>
</tr>
<tr>
<td>REPEAT ALL 3 4 0 14</td>
</tr>
</tbody>
</table>

For joints 1 through 6, the joint number followed by the X, Y and Z coordinates are specified first.

Next, using the coordinates of joints 11 and 25 as the basis, joints 12 through 24 are generated using linear interpolation.

Following this, using the data of these 21 joints (1 through 6 and 11 through 25), 63 new joints are generated. To achieve this, the X coordinate of these 21 joints is incremented by 4 meters and the Z coordinate is incremented by 14 meters, in 3 successive operations.

The REPEAT ALL command is used for the generation. Details of this command is available in TR.11 Joint Coordinates Specification (on page 2425). The results of the generation may be visually verified using STAAD.Pro's graphical viewing facilities.

<table>
<thead>
<tr>
<th>MEMBER INCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23</td>
</tr>
<tr>
<td>26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44</td>
</tr>
<tr>
<td>47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65</td>
</tr>
<tr>
<td>68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86</td>
</tr>
</tbody>
</table>

The member connectivity data (joint numbers between which members are connected) is specified for the 24 columns for the structure. The above method, where the member number is followed by the 2 node numbers, is the explicit definition method. No generation is involved here.

| 101 11 12 114 |
| 202 32 33 215 |
| 303 53 54 316 |
| 404 74 75 417 |

The member connectivity data is specified for the pier cap beams for the structure. The above method is a combination of explicit definition and generation. For example, member 101 is defined as connected between 11 & 12. Then, by incrementing those nodes by 1 unit at a time (which is the default increment), the incidences of members 102 to 114 are generated. Similarly, we create members 202 to 215, 303 to 316, and, 404 to 417.

<table>
<thead>
<tr>
<th>DEFINE MESH</th>
</tr>
</thead>
<tbody>
<tr>
<td>A JOINT 11</td>
</tr>
<tr>
<td>B JOINT 25</td>
</tr>
<tr>
<td>C JOINT 46</td>
</tr>
<tr>
<td>D JOINT 32</td>
</tr>
<tr>
<td>E JOINT 67</td>
</tr>
<tr>
<td>F JOINT 53</td>
</tr>
<tr>
<td>G JOINT 88</td>
</tr>
<tr>
<td>H JOINT 74</td>
</tr>
</tbody>
</table>

The next step is to generate the deck slab which will be modeled using plate elements. For this, we use a technique called mesh generation. Mesh generation is a process of generating several "child" elements from a "parent" or "super" element. The above set of commands defines the corner nodes of the super-element. Details of the above can be found in Section 5.14 of the Technical Reference manual.

Note that instead of elaborately defining the coordinates of the corner nodes of the super-elements, we have taken advantage of the fact that the coordinates of these joints (A through H) have already been defined or
generated earlier. Thus, A is the same as joint 11 while D is the same as joint 32. Alternatively, we could have defined the super-element nodes as A -1 10 0 ; B 16.5 10 0 ; C 20.5 10 14 ; D 3 10 14 ; etc.

| GENERATE ELEMENT | MESH ABCD 14 12 | MESH DCEF 14 12 | MESH FEGH 14 12 |

The above lines are the instructions for generating the “child” elements from the super-elements. For example, from the super-element bound by the corners A, B, C and D (which in turn are nodes 11, 25, 46 and 32), we generate a total of 14X12=168 elements, with 14 divisions along the edges AB and CD, and 12 along the edges BC and DA. These are the elements which make up the first span.

Similarly, 168 elements are created for the 2nd span, and another 168 for the 3rd span.

It may be noted here that we have taken great care to ensure that the resulting elements and the piercap beams form a perfect fit. In other words, there is no overlap between the two in a manner that nodes of the beams are at a different point in space than nodes of elements. At every node along their common boundary, plates and beams are properly connected. This is absolutely essential to ensure proper transfer of load and stiffness from beams to plates and vice versa. The tools of the graphical user interface may be used to confirm that beam-plate connectivity is proper for this model.

| START GROUP DEFINITION |
| MEMBER |
| _GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417 |
| _PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73 |
| ELEMENT |
| _P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471 |
| _P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555 |
| _P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639 |
| _P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737 |
| _P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807 |
| _P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905 |
| END GROUP DEFINITION |

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the piercap beams are being grouped to a name called GIRDERS, the pier columns are assigned the name PIERS, and so on. For the deck, a few selected elements are chosen into a few selective groups. The reason is that these elements happen to be right beneath wheels of vehicles whose weight will be used in the frequency calculation.

| MEMBER PROPERTY |
| _GIRDERS PRIS YD 0.6 ZD 0.6 |
| _PIERS PRIS YD 1.0 |

Member properties are assigned as prismatic rectangular sections for the girders, and prismatic circular sections for the columns.

| ELEMENT PROPERTY |
| YRA 9 11 TH 0.375 |

The plate elements of the deck slab, which happen to be at a Y elevation of 10 metres (between a YRANGE of 9 metres and 11 metres) are assigned a thickness of 375 mms.

| UNIT KNS MMS |
| DEFINE MATERIAL START |
| ISOTROPIC CONCRETE |
| E 21.0 |
| POISSON 0.17 |
| DENSITY 2.36158e-008 |
The Modulus of elasticity (E) is set to 21000 N/sq.mm for all members. The keyword CONSTANTS has to precede this data. Built-in default value for Poisson's ratio for concrete is also assigned to ALL members and elements.

Following a change of units, density of concrete is specified.

The base nodes of the piers are fully restrained (FIXED supports).

Theoretically, a structure has as many modes of vibration as the number of degrees of freedom in the model. However, the limitations of the mathematical process used in extracting modes may limit the number of modes that can actually be extracted. In a large structure, the extraction process can also be very time consuming. Further, not all modes are of equal importance. (One measure of the importance of modes is the participation factor of that mode.) In many cases, the first few modes may be sufficient to obtain a significant portion of the total dynamic response.

Due to these reasons, in the absence of any explicit instruction, STAAD calculates only the first 6 modes. This is like saying that the command CUT OFF MODE SHAPE 6 has been specified. (Versions of STAAD prior to STAAD.Pro 2000 calculated only 3 modes by default).

If the inspection of the first 6 modes reveals that the overall vibration pattern of the structure has not been obtained, one may ask STAAD to compute a larger (or smaller) number of modes with the help of this command. The number that follows this command is the number of modes being requested. In our example, we are asking for 65 modes by specifying CUT OFF MODE SHAPE 65.
The mathematical method that STAAD uses is called the eigen extraction method. Some information on this is available in G.17.3 Dynamic Analysis (on page 2362). The method involves 2 matrices - the stiffness matrix, and the mass matrix.

The stiffness matrix, usually called the \([K]\) matrix, is assembled using data such as member and element lengths, member and element properties, modulus of elasticity, Poisson’s ratio, member and element releases, member offsets, support information, etc.

For assembling the mass matrix, called the \([M]\) matrix, STAAD uses the load data specified in the load case in which the MODAL CAL REQ command is specified. So, some of the important aspects to bear in mind are:

1. The input you specify is weights, not masses. Internally, STAAD will convert weights to masses by dividing the input by "g", the acceleration due to gravity.
2. If the structure is declared as a PLANE frame, there are 2 possible directions of vibration - global X, and global Y. If the structure is declared as a SPACE frame, there are 3 possible directions - global X, global Y and global Z. However, this does not guarantee that STAAD will automatically consider the masses for vibration in all the available directions.

You have control over and are responsible for specifying the directions in which the masses ought to vibrate. In other words, if a weight is not specified along a certain direction, the corresponding degrees of freedom (such as for example, global X at node 34 hypothetically) will not receive a contribution in the mass matrix. The mass matrix is assembled using only the masses from the weights and directions you specify.

In our example, notice that we are specifying the selfweight along global X, Y and Z directions. Similarly, a 200 kg/sq.m pressure load is also specified along all 3 directions on the deck.

But for the truck loads, we choose to apply it on just a few elements in the global Y and Z directions only. The reasoning is something like - for the X direction, the mass is not capable of vibrating because the tires allow the truck to roll along X. Remember, this is just a demonstration example, not necessarily what you may want to do.

The point we want to illustrate is that if a user wants to restrict a certain weight to certain directions only, all he/she has to do is not provide the directions in which those weights cannot vibrate in.

3. As much as possible, provide absolute values for the weights. STAAD is programmed to algebraically add the weights at nodes. So, if some weights are specified as positive numbers and others as negative, the total weight at a given node is the algebraic summation of all the weights in the global directions at that node and the mass is then derived from this algebraic resultant.

This is the command which tells the program that frequencies and modes should be calculated. It is specified inside a load case. In other words, this command accompanies the loads that are to be used in generating the mass matrix.

Frequencies and modes have to be calculated also when dynamic analysis such as response spectrum or time history analysis is carried out. But in such analyses, the MODAL CALCULATION REQUESTED command is not explicitly required. When STAAD encounters the command for response spectrum (see example 11) and time history (see examples 16 and 22), it automatically will carry out a frequency extraction without the help of the MODAL.. command.
This initiates the processes which are required to obtain the frequencies. Frequencies, periods and participation factors are automatically reported in the output file when the operation is completed.

**FINISH**

This terminates the STAAD run.

**Input File**

```
STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
IGNORE LIST
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;
11 -1 10 0 25 16.5 10 0
REPEAT ALL 3 4 0 14
MEMBER INCI
1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86
101 11 12 114
202 32 33 215
303 53 54 316
404 74 75 417
DEFINE MESH
A JOINT 11
B JOINT 25
C JOINT 46
D JOINT 32
E JOINT 67
F JOINT 53
G JOINT 88
H JOINT 74
GENERATE ELEMENT
MESH ABCD 14 12
MESH DCEF 14 12
MESH FEGH 14 12
START GROUP DEFINITION
MEMBER
_GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
_PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
ELEMENT
_P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
_P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
_P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
_P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
_P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
_P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
END GROUP DEFINITION
MEMBER PROPERTY
_GIRDERS PRIS YD 0.6 ZD 0.6
_PIERS PRIS YD 1.0
ELEMENT PROPERTY
YRA 9 11 TH 0.375
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
```
POISSON 0.17
DENSITY 2.4e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
CUT OFF MODE SHAPE 65
UNIT KGS METER
LOAD 1 FREQUENCY CALCULATION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
* PERMANENT WEIGHTS ON DECK
ELEMENT LOAD
YRA 9 11 PR GX 200
YRA 9 11 PR GY 200
YRA 9 11 PR GZ 200
* VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
ELEMENT LOAD
_P1 PR GY 700
_P2 PR GY 700
_P3 PR GY 700
_P4 PR GY 700
_P5 PR GY 700
_P6 PR GY 700
_P1 PR GZ 700
_P2 PR GZ 700
_P3 PR GZ 700
_P4 PR GZ 700
_P5 PR GZ 700
_P6 PR GZ 700
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH

STAAD Output File

******************************************************************************
* STAAD.Pro CONNECT Edition                                           *
* Version 22.01.00.**                                                 *
* Proprietary Program of                                               *
* Bentley Systems, Inc.                                               *
* Date= APR 14, 2019                                                   *
* Time= 22:54:17                                                      *
* Licensed to: Bentley Systems Inc                                    *
******************************************************************************

1. STAAD SPACE FREQUENCIES OF VIBRATION OF A SKewed BRIDGE
INPUT FILE: UK-28 Calculation of Modes and Frequencies of a Bridge.STD
2. IGNORE LIST
3. UNIT METER KN
4. JOINT COORDINATES
5. 1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0
6. 11 -1 10 0 25 16.5 10 0
7. REPEAT ALL 3 4 0 14
8. MEMBER INCI
9. 1 1 13; 2 2 15; 3 3 17; 4 4 19; 5 5 21; 6 6 23
10. 26 26 34; 27 27 36; 28 28 38; 29 29 40; 30 30 42; 31 31 44
11. 47 47 55; 48 48 57; 49 49 59; 50 50 61; 51 51 63; 52 52 65
12. 68 68 76; 69 69 78; 70 70 80; 71 71 82; 72 72 84; 73 73 86
13. 101 11 12 114
14. 202 32 33 215
15. 303 53 54 316
16. 404 74 75 417
17. DEFINE MESH
18. A JOINT 11
20. C JOINT 46
21. D JOINT 32
22. E JOINT 67
23. F JOINT 53
24. G JOINT 88
25. H JOINT 74
26. GENERATE ELEMENT
27. MESH ABCD 14 12
28. MESH DCEF 14 12
29. MESH FEGH 14 12
30. START GROUP DEFINITION
31. MEMBER
32. _GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
33. _PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
34. ELEMENT
35. _PI 447 TO 450 454 TO 457 461 TO 464 468 TO 471
36. _P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
37. _P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
38. _P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
39. FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
40. END GROUP DEFINITION
41. MEMBER PROPERTY
42. _GIRDERS PRIS YD 0.6 ZD 0.6
43. _PIERS PRIS YD 1.0
44. ELEMENT PROPERTY
45. YRA 9 11 TH 0.375
46. UNITS MMS
47. DEFINE MATERIAL START
48. ISOTROPIC CONCRETE
49. E 21.0
50. POISSON 0.17
51. DENSITY 2.4E-008
52. ALPHA 5E-006
53. DAMP 0.05
54. G 9.25
55. TYPE CONCRETE
56. STRENGTH FCU 0.0275
57. END DEFINE MATERIAL
58. CONSTANTS
60. MATERIAL CONCRETE ALL
61. SUPPORTS
62. 1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
63. CUT OFF MODE SHAPE 65
64. UNIT KGS METER
65. LOAD 1 FREQUENCY CALCULATION
66. SELFWEIGHT X 1.0
67. SELFWEIGHT Y 1.0
68. SELFWEIGHT Z 1.0
69. * PERMANENT WEIGHTS ON DECK
70. ELEMENT LOAD
71. YRA 9 11 PR GX 200
72. YRA 9 11 PR GY 200
73. YRA 9 11 PR GZ 200
74. * VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
75. ELEMENT LOAD
76. _P1 PR GY 700
77. _P2 PR GY 700
78. _P3 PR GY 700
79. _P4 PR GY 700
80. _P5 PR GY 700
81. _P6 PR GY 700
82. _P1 PR GZ 700
83. _P2 PR GZ 700
84. _P3 PR GZ 700
85. _P4 PR GZ 700
86. _P5 PR GZ 700
87. _P6 PR GZ 700
88. MODAL CALCULATION REQUESTED
89. PERFORM ANALYSIS

FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE -- PAGE NO. 3

Problem Statistics
-----------------------------------
NUMBER OF JOINTS        579  NUMBER OF MEMBERS      80
NUMBER OF PLATES        504  NUMBER OF SOLIDS        0
NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     24

Using 64-bit analysis engine.
SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =    3330
TOTAL LOAD COMBINATION CASES =     0  SO FAR.
** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS ARE IGNORED. ELEM.NO. 101
** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS ARE IGNORED. ELEM.NO. 101
** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS ARE IGNORED. ELEM.NO. 101
***NOTE: MASSES DEFINED UNDER LOAD#       1 WILL FORM THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.

Eigen Method : Subspace
-------------------------
NUMBER OF MODES REQUESTED              =    65
NUMBER OF EXISTING MASSES IN THE MODEL =  1665
NUMBER OF MODES THAT WILL BE USED      =    65
*** Eigensolution : Advanced Method ***

FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE -- PAGE NO. 4

CALCULATED FREQUENCIES FOR LOAD CASE 1

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.637</td>
<td>0.61071</td>
</tr>
<tr>
<td>MODE</td>
<td>FREQUENCY (CYCLES/SEC)</td>
<td>PERIOD (SEC)</td>
</tr>
<tr>
<td>------</td>
<td>------------------------</td>
<td>--------------</td>
</tr>
<tr>
<td>2</td>
<td>2.604</td>
<td>0.38399</td>
</tr>
<tr>
<td>3</td>
<td>2.886</td>
<td>0.34652</td>
</tr>
<tr>
<td>4</td>
<td>3.760</td>
<td>0.26597</td>
</tr>
<tr>
<td>5</td>
<td>4.083</td>
<td>0.24494</td>
</tr>
<tr>
<td>6</td>
<td>4.396</td>
<td>0.22747</td>
</tr>
<tr>
<td>7</td>
<td>4.533</td>
<td>0.22062</td>
</tr>
<tr>
<td>8</td>
<td>4.696</td>
<td>0.21293</td>
</tr>
<tr>
<td>9</td>
<td>5.049</td>
<td>0.19806</td>
</tr>
<tr>
<td>10</td>
<td>7.232</td>
<td>0.13828</td>
</tr>
<tr>
<td>11</td>
<td>7.281</td>
<td>0.13734</td>
</tr>
<tr>
<td>12</td>
<td>7.407</td>
<td>0.13501</td>
</tr>
<tr>
<td>13</td>
<td>10.360</td>
<td>0.09653</td>
</tr>
<tr>
<td>14</td>
<td>10.758</td>
<td>0.09296</td>
</tr>
<tr>
<td>15</td>
<td>11.195</td>
<td>0.08932</td>
</tr>
<tr>
<td>16</td>
<td>11.310</td>
<td>0.08841</td>
</tr>
<tr>
<td>17</td>
<td>11.605</td>
<td>0.08617</td>
</tr>
<tr>
<td>18</td>
<td>11.872</td>
<td>0.08423</td>
</tr>
<tr>
<td>19</td>
<td>11.956</td>
<td>0.08364</td>
</tr>
<tr>
<td>20</td>
<td>12.134</td>
<td>0.08241</td>
</tr>
<tr>
<td>21</td>
<td>12.542</td>
<td>0.08194</td>
</tr>
<tr>
<td>22</td>
<td>13.727</td>
<td>0.07285</td>
</tr>
<tr>
<td>23</td>
<td>14.719</td>
<td>0.06794</td>
</tr>
<tr>
<td>24</td>
<td>14.831</td>
<td>0.06743</td>
</tr>
<tr>
<td>25</td>
<td>15.202</td>
<td>0.06578</td>
</tr>
<tr>
<td>26</td>
<td>17.387</td>
<td>0.05751</td>
</tr>
<tr>
<td>27</td>
<td>17.560</td>
<td>0.05695</td>
</tr>
<tr>
<td>28</td>
<td>17.828</td>
<td>0.05609</td>
</tr>
<tr>
<td>29</td>
<td>19.795</td>
<td>0.05852</td>
</tr>
<tr>
<td>30</td>
<td>19.988</td>
<td>0.05803</td>
</tr>
<tr>
<td>31</td>
<td>20.596</td>
<td>0.05523</td>
</tr>
<tr>
<td>32</td>
<td>20.688</td>
<td>0.05496</td>
</tr>
<tr>
<td>33</td>
<td>20.901</td>
<td>0.05471</td>
</tr>
<tr>
<td>34</td>
<td>21.212</td>
<td>0.05412</td>
</tr>
<tr>
<td>35</td>
<td>21.502</td>
<td>0.05361</td>
</tr>
<tr>
<td>36</td>
<td>21.873</td>
<td>0.05302</td>
</tr>
<tr>
<td>37</td>
<td>22.161</td>
<td>0.05262</td>
</tr>
<tr>
<td>38</td>
<td>23.248</td>
<td>0.05101</td>
</tr>
<tr>
<td>39</td>
<td>23.593</td>
<td>0.05061</td>
</tr>
<tr>
<td>40</td>
<td>24.730</td>
<td>0.04804</td>
</tr>
<tr>
<td>41</td>
<td>24.730</td>
<td>0.04804</td>
</tr>
<tr>
<td>42</td>
<td>25.535</td>
<td>0.04391</td>
</tr>
<tr>
<td>43</td>
<td>26.128</td>
<td>0.03828</td>
</tr>
<tr>
<td>44</td>
<td>26.537</td>
<td>0.03768</td>
</tr>
<tr>
<td>45</td>
<td>26.938</td>
<td>0.03712</td>
</tr>
<tr>
<td>46</td>
<td>27.433</td>
<td>0.03645</td>
</tr>
<tr>
<td>47</td>
<td>27.884</td>
<td>0.03586</td>
</tr>
<tr>
<td>48</td>
<td>29.090</td>
<td>0.03438</td>
</tr>
<tr>
<td>49</td>
<td>29.686</td>
<td>0.03369</td>
</tr>
<tr>
<td>50</td>
<td>29.927</td>
<td>0.03341</td>
</tr>
<tr>
<td>51</td>
<td>31.110</td>
<td>0.03214</td>
</tr>
</tbody>
</table>

**Frequencies of Vibration of a Skewed Bridge**

**Calculated Frequencies for Load Case 1**

--- PAGE NO. 5 ---
<table>
<thead>
<tr>
<th>MODE</th>
<th>MODAL X</th>
<th>MODAL Y</th>
<th>MODAL Z</th>
<th>MODAL WEIGHT (MODAL MASS TIMES g) IN KGS</th>
<th>GENERALIZED MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.313672E+02</td>
<td>1.108398E+00</td>
<td>1.201129E+06</td>
<td>1.221103E+06</td>
<td>1.221103E+06</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.105162E+06</td>
<td>2.912622E+00</td>
<td>2.323269E+02</td>
<td>1.098762E+06</td>
<td>1.098762E+06</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2.073649E-01</td>
<td>2.786898E+03</td>
<td>2.769101E+00</td>
<td>5.493268E+05</td>
<td>5.493268E+05</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1.462758E+00</td>
<td>3.955818E+04</td>
<td>6.224106E+00</td>
<td>2.014392E+05</td>
<td>2.014392E+05</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>5.986044E+02</td>
<td>6.953900E+02</td>
<td>2.771716E+02</td>
<td>9.013609E+04</td>
<td>9.013609E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1.854187E+01</td>
<td>3.566982E+00</td>
<td>3.225212E+00</td>
<td>7.500848E+04</td>
<td>7.500848E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>5.356242E+00</td>
<td>2.742697E+00</td>
<td>8.417095E+00</td>
<td>7.761683E+04</td>
<td>7.761683E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>1.310402E+02</td>
<td>1.245065E+01</td>
<td>1.558885E+02</td>
<td>8.975814E+04</td>
<td>8.975814E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>9.357269E+00</td>
<td>1.087438E+01</td>
<td>2.082024E+03</td>
<td>4.957080E+04</td>
<td>4.957080E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1.310402E+02</td>
<td>1.245065E+01</td>
<td>1.558885E+02</td>
<td>8.975814E+04</td>
<td>8.975814E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>5.356242E+00</td>
<td>2.742697E+00</td>
<td>8.417095E+00</td>
<td>7.761683E+04</td>
<td>7.761683E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>1.854187E+01</td>
<td>3.566982E+00</td>
<td>3.225212E+00</td>
<td>7.500848E+04</td>
<td>7.500848E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>2.411181E+00</td>
<td>1.344846E+04</td>
<td>2.572347E-01</td>
<td>5.878995E+04</td>
<td>5.878995E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>6.757432E+02</td>
<td>6.222985E+03</td>
<td>5.878995E+04</td>
<td>3.533635E+04</td>
<td>3.533635E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>3.712278E+00</td>
<td>4.299742E+04</td>
<td>5.310814E+00</td>
<td>3.190311E+04</td>
<td>3.190311E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>2.171634E+02</td>
<td>6.963874E+01</td>
<td>2.873028E+00</td>
<td>7.802286E+02</td>
<td>7.802286E+02</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>2.478604E+01</td>
<td>4.222519E+04</td>
<td>2.375592E-03</td>
<td>3.571038E+04</td>
<td>3.571038E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>9.426945E+00</td>
<td>4.274711E+00</td>
<td>5.068050E-01</td>
<td>6.420727E+04</td>
<td>6.420727E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1.318833E+01</td>
<td>1.391960E+02</td>
<td>9.997939E+00</td>
<td>3.683261E+04</td>
<td>3.683261E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>2.655567E+00</td>
<td>5.108468E+03</td>
<td>2.802262E+00</td>
<td>4.277587E+04</td>
<td>4.277587E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>3.666383E+00</td>
<td>4.719298E+00</td>
<td>1.036419E+00</td>
<td>3.639540E+04</td>
<td>3.639540E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>1.618305E+00</td>
<td>2.671420E+04</td>
<td>8.032944E-01</td>
<td>3.347565E+04</td>
<td>3.347565E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>1.289408E+00</td>
<td>8.348047E+01</td>
<td>2.136199E-02</td>
<td>1.948813E+04</td>
<td>1.948813E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>1.096810E-02</td>
<td>3.990261E+00</td>
<td>4.500325E-01</td>
<td>1.839953E+04</td>
<td>1.839953E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>2.516875E-03</td>
<td>2.833839E-03</td>
<td>1.672893E-04</td>
<td>4.283791E+05</td>
<td>4.283791E+05</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>1.800354E+00</td>
<td>1.729541E+03</td>
<td>1.115994E+01</td>
<td>4.982347E+04</td>
<td>4.982347E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>3.094704E-01</td>
<td>5.689236E+04</td>
<td>1.057619E-01</td>
<td>7.138174E+04</td>
<td>7.138174E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>4.237568E+00</td>
<td>7.117003E+03</td>
<td>1.789099E+00</td>
<td>7.941099E+04</td>
<td>7.941099E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>6.985918E-03</td>
<td>7.120621E+00</td>
<td>2.285432E-02</td>
<td>7.795511E+04</td>
<td>7.795511E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>1.000846E-02</td>
<td>2.815537E+00</td>
<td>8.068273E-04</td>
<td>1.854435E+05</td>
<td>1.854435E+05</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>3.867976E-01</td>
<td>1.895886E+03</td>
<td>9.105723E-03</td>
<td>2.331780E+04</td>
<td>2.331780E+04</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MODE</td>
<td>X</td>
<td>Y</td>
<td>Z</td>
<td>SUMM-X</td>
<td>SUMM-Y</td>
<td>SUMM-Z</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>-------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.01</td>
<td>0.00</td>
<td>99.04</td>
<td>0.012</td>
<td>0.00</td>
<td>99.043</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>99.14</td>
<td>0.00</td>
<td>0.02</td>
<td>99.154</td>
<td>0.00</td>
<td>99.063</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.23</td>
<td>0.00</td>
<td>99.154</td>
<td>0.230</td>
<td>99.063</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>3.26</td>
<td>0.00</td>
<td>99.154</td>
<td>3.492</td>
<td>99.063</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.04</td>
<td>0.05</td>
<td>99.154</td>
<td>3.532</td>
<td>99.113</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.05</td>
<td>0.06</td>
<td>0.02</td>
<td>99.204</td>
<td>3.589</td>
<td>99.136</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>29.41</td>
<td>0.00</td>
<td>99.206</td>
<td>33.002</td>
<td>99.136</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>22.62</td>
<td>0.00</td>
<td>99.206</td>
<td>55.618</td>
<td>99.137</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>0.00</td>
<td>0.14</td>
<td>0.19</td>
<td>99.736</td>
<td>55.758</td>
<td>99.323</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.00</td>
<td>0.13</td>
<td>0.00</td>
<td>99.736</td>
<td>55.884</td>
<td>99.323</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.00</td>
<td>0.05</td>
<td>0.00</td>
<td>99.737</td>
<td>55.939</td>
<td>99.323</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>99.737</td>
<td>55.974</td>
<td>99.323</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>0.00</td>
<td>0.00</td>
<td>0.57</td>
<td>99.740</td>
<td>55.974</td>
<td>99.892</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>0.00</td>
<td>0.01</td>
<td>0.01</td>
<td>99.740</td>
<td>55.985</td>
<td>99.901</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>0.00</td>
<td>0.36</td>
<td>0.01</td>
<td>99.740</td>
<td>56.344</td>
<td>99.911</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>0.00</td>
<td>0.01</td>
<td>0.01</td>
<td>99.741</td>
<td>56.350</td>
<td>99.925</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.742</td>
<td>56.350</td>
<td>99.925</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>0.01</td>
<td>0.00</td>
<td>0.01</td>
<td>99.753</td>
<td>56.351</td>
<td>99.938</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.754</td>
<td>56.354</td>
<td>99.938</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>99.754</td>
<td>56.360</td>
<td>99.940</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.754</td>
<td>56.360</td>
<td>99.940</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.766</td>
<td>56.360</td>
<td>99.940</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.766</td>
<td>56.364</td>
<td>99.947</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>0.00</td>
<td>0.02</td>
<td>0.00</td>
<td>99.766</td>
<td>56.380</td>
<td>99.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>0.00</td>
<td>0.00</td>
<td>0.02</td>
<td>99.767</td>
<td>56.381</td>
<td>99.965</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>0.00</td>
<td>0.19</td>
<td>0.00</td>
<td>99.767</td>
<td>56.568</td>
<td>99.965</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>99.768</td>
<td>56.597</td>
<td>99.965</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>0.00</td>
<td>1.11</td>
<td>0.00</td>
<td>99.768</td>
<td>57.705</td>
<td>99.965</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>0.06</td>
<td>0.51</td>
<td>0.00</td>
<td>99.829</td>
<td>58.218</td>
<td>99.978</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>0.00</td>
<td>3.55</td>
<td>0.00</td>
<td>99.832</td>
<td>61.763</td>
<td>99.978</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>0.02</td>
<td>0.02</td>
<td>0.00</td>
<td>99.852</td>
<td>61.785</td>
<td>99.971</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>0.00</td>
<td>3.48</td>
<td>0.00</td>
<td>99.854</td>
<td>65.267</td>
<td>99.971</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>0.00</td>
<td>0.17</td>
<td>0.00</td>
<td>99.855</td>
<td>65.438</td>
<td>99.971</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>99.856</td>
<td>65.449</td>
<td>99.971</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Understanding the output

After the analysis is complete, look at the output file. (This file can be viewed by selecting the **Analysis Output** tool in the **View** group on the **Utilities** ribbon tab).

1. Mode number and corresponding frequencies and periods

---

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Summ-X</th>
<th>Summ-Y</th>
<th>Summ-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>35</td>
<td>0.00</td>
<td>0.42</td>
<td>0.00</td>
<td>99.856</td>
<td>65.871</td>
<td>99.972</td>
</tr>
<tr>
<td>36</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.856</td>
<td>65.871</td>
<td>99.972</td>
</tr>
<tr>
<td>37</td>
<td>0.00</td>
<td>2.20</td>
<td>0.00</td>
<td>99.856</td>
<td>68.074</td>
<td>99.972</td>
</tr>
<tr>
<td>38</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>99.856</td>
<td>68.081</td>
<td>99.972</td>
</tr>
<tr>
<td>39</td>
<td>0.00</td>
<td>0.03</td>
<td>0.00</td>
<td>99.856</td>
<td>68.114</td>
<td>99.972</td>
</tr>
<tr>
<td>40</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.856</td>
<td>68.116</td>
<td>99.972</td>
</tr>
<tr>
<td>41</td>
<td>0.00</td>
<td>0.14</td>
<td>0.00</td>
<td>99.857</td>
<td>68.259</td>
<td>99.973</td>
</tr>
<tr>
<td>42</td>
<td>0.00</td>
<td>4.69</td>
<td>0.00</td>
<td>99.857</td>
<td>72.950</td>
<td>99.973</td>
</tr>
<tr>
<td>43</td>
<td>0.00</td>
<td>0.06</td>
<td>0.00</td>
<td>99.857</td>
<td>73.008</td>
<td>99.973</td>
</tr>
<tr>
<td>44</td>
<td>0.00</td>
<td>0.06</td>
<td>0.00</td>
<td>99.857</td>
<td>73.067</td>
<td>99.973</td>
</tr>
<tr>
<td>45</td>
<td>0.00</td>
<td>0.02</td>
<td>0.00</td>
<td>99.857</td>
<td>73.090</td>
<td>99.973</td>
</tr>
<tr>
<td>46</td>
<td>0.00</td>
<td>0.16</td>
<td>0.00</td>
<td>99.857</td>
<td>73.247</td>
<td>99.973</td>
</tr>
<tr>
<td>47</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.857</td>
<td>73.247</td>
<td>99.973</td>
</tr>
<tr>
<td>48</td>
<td>0.00</td>
<td>0.38</td>
<td>0.00</td>
<td>99.858</td>
<td>73.622</td>
<td>99.973</td>
</tr>
<tr>
<td>49</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.858</td>
<td>73.624</td>
<td>99.973</td>
</tr>
<tr>
<td>50</td>
<td>0.00</td>
<td>0.14</td>
<td>0.00</td>
<td>99.858</td>
<td>73.624</td>
<td>99.973</td>
</tr>
<tr>
<td>51</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.858</td>
<td>73.624</td>
<td>99.973</td>
</tr>
<tr>
<td>52</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.858</td>
<td>73.624</td>
<td>99.973</td>
</tr>
<tr>
<td>53</td>
<td>0.00</td>
<td>0.10</td>
<td>0.00</td>
<td>99.859</td>
<td>73.738</td>
<td>99.973</td>
</tr>
<tr>
<td>54</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>99.860</td>
<td>73.743</td>
<td>99.974</td>
</tr>
<tr>
<td>55</td>
<td>0.00</td>
<td>0.02</td>
<td>0.00</td>
<td>99.860</td>
<td>73.765</td>
<td>99.974</td>
</tr>
<tr>
<td>56</td>
<td>0.00</td>
<td>0.07</td>
<td>0.00</td>
<td>99.860</td>
<td>73.833</td>
<td>99.974</td>
</tr>
<tr>
<td>57</td>
<td>0.00</td>
<td>0.01</td>
<td>0.00</td>
<td>99.860</td>
<td>73.841</td>
<td>99.974</td>
</tr>
<tr>
<td>58</td>
<td>0.00</td>
<td>0.14</td>
<td>0.00</td>
<td>99.860</td>
<td>73.978</td>
<td>99.975</td>
</tr>
<tr>
<td>59</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>99.861</td>
<td>73.979</td>
<td>99.977</td>
</tr>
<tr>
<td>60</td>
<td>0.00</td>
<td>0.04</td>
<td>0.00</td>
<td>99.861</td>
<td>74.021</td>
<td>99.981</td>
</tr>
<tr>
<td>61</td>
<td>0.00</td>
<td>0.14</td>
<td>0.00</td>
<td>99.866</td>
<td>74.163</td>
<td>99.984</td>
</tr>
<tr>
<td>62</td>
<td>0.00</td>
<td>0.27</td>
<td>0.00</td>
<td>99.868</td>
<td>74.429</td>
<td>99.985</td>
</tr>
<tr>
<td>63</td>
<td>0.00</td>
<td>0.04</td>
<td>0.00</td>
<td>99.868</td>
<td>74.471</td>
<td>99.985</td>
</tr>
<tr>
<td>64</td>
<td>0.00</td>
<td>0.32</td>
<td>0.00</td>
<td>99.868</td>
<td>74.791</td>
<td>99.985</td>
</tr>
<tr>
<td>65</td>
<td>0.00</td>
<td>0.18</td>
<td>0.00</td>
<td>99.869</td>
<td>74.971</td>
<td>99.987</td>
</tr>
<tr>
<td>66</td>
<td>0.00</td>
<td>0.07</td>
<td>0.00</td>
<td>99.860</td>
<td>73.833</td>
<td>99.974</td>
</tr>
</tbody>
</table>

---

90. FINISH

---

********** END OF THE STAAD.Pro RUN **********
**** DATE= APR 14, 2019 TIME= 22:54:20 ****

---

* For technical assistance on STAAD.Pro, please visit

* Details about additional assistance from
  * Bentley and Partners can be found at program menu
  * Help->Technical Support

* Copyright (c) 1997-2017 Bentley Systems, Inc.
  * http://www.bentley.com
Since we asked for 65 modes, we obtain a report, a portion of which is as shown:

Table 660: Calculated Frequencies for Load Case 1

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (Cycles/Sec)</th>
<th>Period (Sec)</th>
<th>Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.636</td>
<td>0.61111</td>
<td>1.344E-16</td>
</tr>
<tr>
<td>2</td>
<td>2.602</td>
<td>0.38433</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>3</td>
<td>2.882</td>
<td>0.34695</td>
<td>8.666E-16</td>
</tr>
<tr>
<td>4</td>
<td>3.754</td>
<td>0.26636</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>5</td>
<td>4.076</td>
<td>0.24532</td>
<td>3.466E-16</td>
</tr>
<tr>
<td>6</td>
<td>4.373</td>
<td>0.22870</td>
<td>6.025E-16</td>
</tr>
<tr>
<td>7</td>
<td>4.519</td>
<td>0.22130</td>
<td>5.641E-16</td>
</tr>
<tr>
<td>8</td>
<td>4.683</td>
<td>0.21355</td>
<td>5.253E-16</td>
</tr>
<tr>
<td>9</td>
<td>5.028</td>
<td>0.19889</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>10</td>
<td>7.189</td>
<td>0.13911</td>
<td>8.916E-16</td>
</tr>
<tr>
<td>11</td>
<td>7.238</td>
<td>0.13815</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>12</td>
<td>7.363</td>
<td>0.13582</td>
<td>0.000E+00</td>
</tr>
</tbody>
</table>

Table 661: Mass Participation Factors in Percent

<table>
<thead>
<tr>
<th>Mode</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>ΣX</th>
<th>ΣY</th>
<th>ΣZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.01</td>
<td>0.00</td>
<td>99.04</td>
<td>0.012</td>
<td>0.000</td>
<td>99.042</td>
</tr>
<tr>
<td>2</td>
<td>99.14</td>
<td>0.00</td>
<td>0.02</td>
<td>99.151</td>
<td>0.000</td>
<td>99.061</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.23</td>
<td>0.00</td>
<td>99.151</td>
<td>0.229</td>
<td>99.062</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>3.27</td>
<td>0.00</td>
<td>99.151</td>
<td>3.496</td>
<td>99.062</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.04</td>
<td>0.05</td>
<td>99.151</td>
<td>3.536</td>
<td>99.112</td>
</tr>
<tr>
<td>6</td>
<td>0.05</td>
<td>0.04</td>
<td>0.02</td>
<td>99.202</td>
<td>3.575</td>
<td>99.135</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>26.42</td>
<td>0.00</td>
<td>99.204</td>
<td>30.000</td>
<td>99.135</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>25.59</td>
<td>0.00</td>
<td>99.204</td>
<td>55.587</td>
<td>99.136</td>
</tr>
</tbody>
</table>
In the explanation earlier for the CUT OFF MODE command, we said that one measure of the importance of a mode is the participation factor of that mode. We can see from the above report that for vibration along Z direction, the first mode has a 99.04 percent participation. It is also apparent that the 7th mode is primarily a Y direction mode with a 26.42% participation along Y and 0 in X and Z.

The \( \Sigma X, \Sigma Y \) and \( \Sigma Z \) columns show the cumulative value of the participation of all the modes up to and including a given mode (Corresponding to the SUMM-X, SUMM-Y, and SUMM-Z reported in the output, respectively). One can infer from those terms that if one is interested in 95% participation along X, the first 2 modes are sufficient.

But for the Y direction, even with 10 modes, we barely obtained 60%. The reason for this can be understood by an examination of the nature of the structure. The deck slab is capable of vibrating in several low energy and primarily vertical direction modes. The out-of-plane flexible nature of the slab enables it to vibrate in a manner resembling a series of wave like curves. Masses on either side of the equilibrium point have opposing eigenvector values leading to a lot of cancellation of the contribution from the respective masses. Localized modes, where small pockets in the structure undergo flutter due to their relative weak stiffness compared to the rest of the model, also result in small participation factors.

iii. After the analysis is completed, select Post-processing from the mode menu. This screen contains facilities for graphically examining the shape of the mode in static and animated views. The Dynamics page on the left side of the screen is available for viewing the shape of the mode statically. The Animation option of the Results menu can be used for animating the mode. The mode number can be selected from the Loads and Results tab of the Diagnostics dialog box which opens when the Animation option is chosen. The size to which the mode is drawn is controlled using the Scales tab of the Diagnostics dialog box.

Related Links
• M. To calculate the structure frequency (on page 834)
• TR.34.2 Modal Calculation Command (on page 2791)

EX. UK-29 Time History Analysis of a Frame for Seismic Loads

Analysis and design of a structure for seismic loads is demonstrated in this example. In this model, static load cases are solved along with the seismic load case. For the seismic case, the maximum values of displacements, forces and reactions are obtained. The results of the dynamic case are combined with those of the static cases and steel design is performed on the combined cases.

This problem is installed with the program by default to C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-29 Time History Analysis of a Frame for Seismic Loads.std when you install the program.
Actual input is shown in bold lettering followed by explanation.

**STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS**

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words form a title to identify this project.

**UNIT METER KNS**

The units for the data that follows are specified above.

**JOINT COORDINATES**

1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
REPEAT ALL 1 9.5 0 0
REPEAT ALL 1 0 0 3
17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
REPEAT ALL 1 0 0 3

For joints 1 through 4, the joint number is followed by the X, Y and Z coordinates as specified above. The coordinates of these joints are used as a basis for generating 12 more joints by incrementing the X & Z coordinates by specific amounts. REPEAT ALL commands are used for the generation. Details of these commands are available in Section 5.11 of the Technical Reference manual. Following this, another round of explicit definition (joints 17, 18 & 19) and generation (20, 21 & 22) is carried out. The results of the generation may be visually verified using graphical view features in STAAD.Pro.

**MEMBER INCIDENCES**

1 1 2 3
REPEAT 1 3 4
7 9 10 9
A mixture of explicit definition and generation of member connectivity data (joint numbers between which members are connected) is used to generate 29 members for the structure.

START GROUP DEFINITION
MEMBER
  _VERTICAL 1 TO 12
  _XBEAM 13 TO 20
  _ZBEAM 21 TO 24 29
  _BRACE 25 TO 28
END GROUP DEFINITION

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the columns are being grouped to a name called VERTICAL, the beams running along the X direction are assigned the name XBEAM, and so on.

MEMBER PROPERTIES CANADIAN
  _VERTICAL TA ST W310X97
  _XBEAM TA ST W250X39
  _ZBEAM TA ST C200X17
  _BRACE TA ST L152X152X13

Member properties are assigned from the Canadian steel table. The members which receive these properties are those embedded within the respective group names. The benefit of using the group name is apparent here. Just from the looks of the command, we can understand that the diagonal braces are being assigned a single angle. The alternative, which would be

25 TO 28 TA ST L152X152X13

would have required us to go to the graphical tools to get a sense of what members 25 to 28 are.

UNIT KNS MMS
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 200
  POISSON 0.3
  DENSITY 7.8e-008
  ALPHA 6e-006
  DAMP 0.03
  TYPE STEEL
  STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
  MATERIAL STEEL ALL
  BETA 180 MEMB 21 22

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. The BETA angle for the channels along the left edge is set to 180 so their legs point toward the interior of the structure.
The bottom ends of the columns of the platform are pinned supported.

CUT OFF MODE SHAPE 30

The above command is a critical command if you want to override the default number of modes computed and used in a dynamic analysis. The default, which is 6, may not always be sufficient to capture a significant portion of the structural response in a response spectrum or time history analysis, and hence the need to override the default. This command is explained in Section 5.30 of the Technical Reference manual.

UNIT METER
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the earthquake (ground acceleration) are provided.

Each data set is individually identified by the number that follows the TYPE command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word FORCE that follows the TYPE 1 command signifies that this data set is for a ground acceleration. (If you want to specify a forcing function, the keyword FORCE must be used instead.)

Notice the expression READ EQDATA.TXT. It means that we have chosen to specify the time vs. ground acceleration data in the file called EQDATA.TXT. That file must reside in the same folder as the one in which the data file for this structure resides. As explained in the small examples shown in TR.31.4 Definition of Time History Load (on page 2630), the EQDATA.TXT file is a simple text file containing several pairs of time-acceleration data. A sample portion of that file is as shown below.

<table>
<thead>
<tr>
<th>Time (sec)</th>
<th>Acceleration (m/s²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000</td>
<td>0.006300</td>
</tr>
<tr>
<td>0.0200</td>
<td>0.003640</td>
</tr>
<tr>
<td>0.0400</td>
<td>0.000990</td>
</tr>
<tr>
<td>0.0600</td>
<td>0.004280</td>
</tr>
<tr>
<td>0.0800</td>
<td>0.007580</td>
</tr>
<tr>
<td>0.1000</td>
<td>0.010870</td>
</tr>
</tbody>
</table>

While it may not be apparent from the above numbers, it may also be noted that the geological data for the site the building sits on indicate that the above acceleration values are a fraction of "g", the acceleration due to gravity. Thus, for example, at 0.02 seconds, the acceleration is 0.00364 multiplied by 9.806 m/sec² (or 0.00364 multiplied by 32.2 ft/sec²). Consequently, the burden of informing the program that the values need to be multiplied by "g" is upon us, and we shall do so at a later step.

The arrival time value indicates the relative value of time at which the earthquake begins to act upon the structure. We have chosen 0.0, as there is no other dynamic load on the structure from the relative time standpoint. The modal damping ratio for all the modes is set to 0.05.

LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
SELFWEIGHT Y -1.0

The above data describe a static load case. The selfweight of the structure is acting in the negative global Y direction.

LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY FLOOR LOAD
YRA 6.9 7.1 FLOAD -500
Load case 2 is also a static load case. At the Y=7.0m elevation, our structure has a floor slab. But, it is a non-structural entity which, though capable of carrying the loads acting on itself, is not meant to be an integral part of the framing system. It merely transmits the load to the beam-column grid.

There are uniform area loads on the floor (think of the load as wooden pallets supporting boxes of paper). Since the slab is not part of the structural model, how do we tell the program to transmit the imposed load from the slab to the beams without manually converting them to distributed beam loads ourselves? That is where the floor load utility comes in handy. It is a facility where we specify the load as a pressure, and the program converts the pressure to individual beam loads. Thus, the input required is very simple - load intensity in the form of pressure, and the region of the structure in terms of X, Y and Z coordinates in space, of the area over which the pressure acts.

In the process of converting the pressure to beam loads, STAAD will consider the empty space between criss-crossing beams (in plan view) to be panels, similar to the squares of a chess board. The load on each panel is then transferred to beams surrounding the panel, using a triangular or trapezoidal load distribution method.

| LOAD 3 DYNAMIC LOAD |
| * MASSES |
| SELFWEIGHT X 1.0 |
| SELFWEIGHT Y 1.0 |
| SELFWEIGHT Z 1.0 |
| FLOOR LOAD |
| YRANGE 6.9 7.1 FLOAD 500 GX |
| YRANGE 6.9 7.1 FLOAD 500 GY |
| YRANGE 6.9 7.1 FLOAD 500 GZ |

Load case 3 is the dynamic load case, the one which contains the second part of the instruction set for a dynamic analysis to be performed. The data here are

a. loads which will yield the mass values which will populate the mass matrix
b. the directions of the loads, which will yield the degree of freedom numbers of the mass matrix for being populated.

Thus, the selfweight, as well as the imposed loads on the non-structural slab are to be considered as participating in the vibration along all the global directions.

| GROUND MOTION X 1 1 9.806 |

The above command too is part of load case 3. Here we say that the seismic force, whose characteristics are defined by the TYPE 1 time history input data, acting at arrival time 1, is to be applied along the X direction. We mentioned earlier that the acceleration input data was specified as a fraction of “g”. The number 9.806 indicates the value which the acceleration data, as read from EQDATA.TXT are to be factored by before they are used.

| LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC) |
| 1 1.0 2 1.0 3 1.0 |
| LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC) |
| 1 1.0 2 1.0 3 -1.0 |

In a time history analysis, the member forces FX thru MZ each have a value for every time step. If there are a 1000 time steps, there will be 1000 values of FX, 1000 for FY etc. for that load case. Not all of them can be used in a further calculation like a steel or concrete design. However, the maximum from among those time steps is available. If we want to do a design, one way to make sure that the structure is not under-designed is to create 2 load combination cases involving the dynamic case, a positive combination, and a negative combination.

That is what is being done above. Load combination case no. 11 consists of the sum of the static load cases (1 & 2) with the positive direction of the dynamic load case (3). Load combination case no. 12 consists of the sum of...
the static load cases (1 & 2) with the negative direction of the dynamic load case (3). The user has discretion on what load factors to use with these combinations. We have chosen the factors to be 1.0.

PERFORM ANALYSIS

The above is the instruction to perform the analysis related calculations. That means, computing nodal displacements, support reactions, etc.

PRINT ANALYSIS RESULTS

The above command is an instruction to the program to produce a report of the joint displacements, support reactions and member end forces in the output file. As mentioned earlier, for the dynamic case, these will be just the maximum values, not the ones generated for every time step. If you want to see the results for each time step, you may do so by using STAAD’s Post-processing facilities.

LOAD LIST 11 12
PARAMETER
  CODE CANADIAN
CHECK CODE ALL

A steel design code check is done according to the Canadian code for load cases 11 and 12.

FINISH

Input File

STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
UNIT METER KNS
JOINT COORDINATES
1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
REPEAT ALL 1 9.5 0 0
REPEAT ALL 1 0 0 3
17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
REPEAT ALL 1 0 0 3
MEMBER INCIDENCES
1 1 2 3
REPEAT 1 3 4
7 9 10 9
10 13 14 12
13 4 17; 14 17 18; 15 18 19; 16 19 8
17 12 20; 18 20 21; 19 21 22; 20 22 16
21 2 10; 22 4 12; 23 6 14
24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
START GROUP DEFINITION
MEMBER
  _VERTICAL 1 TO 12
  _XBEAM 13 TO 20
  _ZBEAM 21 TO 24 29
  _BRACE 25 TO 28
END GROUP DEFINITION
MEMBER PROPERTIES CANADIAN
  _VERTICAL TA ST W310X97
  _XBEAM TA ST W250X39
  _ZBEAM TA ST C200X17
  _BRACE TA ST L152X152X13
UNIT KNS MMS
DEFINE MATERIAL START
  ISOTROPIC STEEL
  E 200
  POISSON 0.3
DENSITY 7.8e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 180 MEMB 21 22
SUPPORTS
1 5 9 13 PINNED
CUT OFF MODE SHAPE 30
UNIT KGS METER
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05
LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
SELFWEIGHT Y -1.0
LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY
FLOOR LOAD
YRA 6.9 7.1 FLOAD -500
LOAD 3 DYNAMIC LOAD
* MASSES
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
FLOOR LOAD
YRANGE 6.9 7.1 FLOAD 500 GX
YRANGE 6.9 7.1 FLOAD 500 GY
YRANGE 6.9 7.1 FLOAD 500 GZ
GROUND MOTION X 1 1 9.806
LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
1 1.0 2 1.0 3 1.0
LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
1 1.0 2 1.0 3 -1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 11 12
PARAMETER
CODE CANADA
CHECK CODE ALL
FINISH
1. STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS

INPUT FILE: UK-29 Time History Analysis of a Frame for Seismic Loads.STD

2. UNIT METER KNS

3. JOINT COORDINATES
4. 1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
5. REPEAT ALL 1 9.5 0 0
6. REPEAT ALL 1 0 0 3
7. 17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
8. REPEAT ALL 1 0 0 3

9. MEMBER INCIDENCES
10. 1 1 2 3
11. REPEAT 1 3 4
12. 7 9 10 9
13. 10 13 14 12
14. 13 4 17; 14 17 18; 15 18 19; 16 19 8
15. 17 12 20; 18 20 21; 19 21 22; 20 22 16
16. 21 2 10; 22 4 12; 23 6 14
17. 24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21

18. START GROUP DEFINITION
19. MEMBER
20. _VERTICAL 1 TO 12
21. _XBEAM 13 TO 20
22. _ZBEAM 21 TO 24 29
23. _BRACE 25 TO 28
24. END GROUP DEFINITION
25. MEMBER PROPERTIES CANADIAN
26. _VERTICAL TA ST W310X97
27. _XBEAM TA ST W250X39
28. _ZBEAM TA ST C200X17
29. _BRACE TA ST L152X152X13
30. UNIT KNS MMS

31. DEFINE MATERIAL START
32. ISOTROPIC STEEL
33. E 200
34. POISSON 0.3
35. DENSITY 7.8E-008
36. ALPHA 6E-006
37. DAMP 0.03
38. TYPE STEEL
39. DYNAMIC ANALYSIS FOR SEISMIC LOADS -- PAGE NO. 2

40. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
41. END DEFINE MATERIAL
42. CONSTANTS
43. MATERIAL STEEL ALL
44. BETA 180 MEMB 21 22
45. SUPPORTS
46. 1 5 9 13 PINNED
47. CUT OFF MODE SHAPE 30
48. UNIT KGS METER
49. DEFINE TIME HISTORY
50. TYPE 1 ACCELERATION
51. READ EQDATA.TXT
52. ARRIVAL TIME
53. 0.0
54. DAMPING 0.05
55. LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
55. SELFWEIGHT Y -1.0
56. LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY
57. FLOOR LOAD
58. YRA 6.9 7.1 FLOAD -500
**NOTE** about Floor/OneWay Loads/Weights.
Please note that depending on the shape of the floor you may
have to break up the FLOOR/ONEWAY LOAD into multiple commands.
For details please refer to Technical Reference Manual
Section 5.32.4.2 Note d and/or ".5.32.4.3 Note f.
59. LOAD 3 DYNAMIC LOAD
60. * MASSES
61. SELFWEIGHT X 1.0
62. SELFWEIGHT Y 1.0
63. SELFWEIGHT Z 1.0
64. FLOOR LOAD
65. YRANGE 6.9 7.1 FLOAD 500 GX
66. YRANGE 6.9 7.1 FLOAD 500 GY
67. YRANGE 6.9 7.1 FLOAD 500 GZ
68. GROUND MOTION X 1 1 9.806
69. LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
70. 1 1.0 2 1.0 3 1.0
71. LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
72. 1 1.0 2 1.0 3 -1.0
73. PERFORM ANALYSIS
   DYNAMIC ANALYSIS FOR SEISMIC LOADS
   -- PAGE NO. 3
   PROBLEM STATISTICS
   -----------------------------------
   NUMBER OF JOINTS 22  NUMBER OF MEMBERS 29
   NUMBER OF PLATES  0   NUMBER OF SOLIDS  0
   NUMBER OF SURFACES 0   NUMBER OF SUPPORTS 4
   Using 64-bit analysis engine.
   SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL PRIMARY LOAD CASES = 3, TOTAL DEGREES OF FREEDOM = 120
   TOTAL LOAD COMBINATION CASES = 2 So Far.
   ***NOTE: MASSES DEFINED UNDER LOAD# 3 WILL FORM
   THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
   EIGEN METHOD : SUBSPACE
   -----------------------------------
   NUMBER OF MODES REQUESTED = 30
   NUMBER OF EXISTING MASSES IN THE MODEL = 54
   NUMBER OF MODES THAT WILL BE USED = 30
   *** EIGENSOLUTION : ADVANCED METHOD ***
   DYNAMIC ANALYSIS FOR SEISMIC LOADS
   -- PAGE NO. 4
   CALCULATED FREQUENCIES FOR LOAD CASE 3
<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY(CYCLES/SEC)</th>
<th>PERIOD(SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.693</td>
<td>1.44295</td>
</tr>
<tr>
<td>2</td>
<td>1.215</td>
<td>0.82296</td>
</tr>
<tr>
<td>3</td>
<td>1.365</td>
<td>0.73265</td>
</tr>
<tr>
<td>4</td>
<td>1.561</td>
<td>0.64059</td>
</tr>
<tr>
<td>5</td>
<td>2.077</td>
<td>0.48142</td>
</tr>
<tr>
<td>6</td>
<td>3.044</td>
<td>0.32846</td>
</tr>
<tr>
<td>7</td>
<td>4.217</td>
<td>0.23712</td>
</tr>
<tr>
<td>8</td>
<td>4.273</td>
<td>0.23404</td>
</tr>
<tr>
<td>9</td>
<td>5.538</td>
<td>0.18058</td>
</tr>
<tr>
<td>10</td>
<td>5.543</td>
<td>0.18039</td>
</tr>
<tr>
<td>11</td>
<td>5.728</td>
<td>0.17457</td>
</tr>
<tr>
<td>12</td>
<td>12.732</td>
<td>0.07854</td>
</tr>
<tr>
<td>13</td>
<td>12.741</td>
<td>0.07849</td>
</tr>
</tbody>
</table>
### MODAL WEIGHT (MODAL MASS TIMES g) IN KGS

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.330908E-16</td>
<td>3.856561E-17</td>
<td>1.501941E+04</td>
<td>8.845560E+03</td>
</tr>
<tr>
<td>2</td>
<td>1.725302E+04</td>
<td>4.369968E-02</td>
<td>2.140145E-16</td>
<td>1.677679E+04</td>
</tr>
<tr>
<td>3</td>
<td>2.909841E-11</td>
<td>5.820389E-16</td>
<td>8.419906E-01</td>
<td>1.266359E+04</td>
</tr>
<tr>
<td>4</td>
<td>8.919546E-12</td>
<td>1.342864E-15</td>
<td>4.822829E+00</td>
<td>2.596549E+04</td>
</tr>
<tr>
<td>5</td>
<td>5.385775E-16</td>
<td>1.266211E-18</td>
<td>2.369262E+03</td>
<td>5.207686E+03</td>
</tr>
<tr>
<td>6</td>
<td>2.267313E-29</td>
<td>3.707073E-30</td>
<td>8.107877E-14</td>
<td>5.200768E+03</td>
</tr>
<tr>
<td>7</td>
<td>6.805920E-30</td>
<td>3.131805E-13</td>
<td>5.977653E-13</td>
<td>5.200768E+03</td>
</tr>
<tr>
<td>8</td>
<td>3.523092E-16</td>
<td>2.630816E-16</td>
<td>1.923070E+00</td>
<td>5.200768E+03</td>
</tr>
<tr>
<td>9</td>
<td>6.281204E-03</td>
<td>9.100303E+03</td>
<td>1.326257E-10</td>
<td>6.018268E+03</td>
</tr>
<tr>
<td>10</td>
<td>3.796626E-12</td>
<td>5.397674E-06</td>
<td>2.331646E-01</td>
<td>6.033971E+03</td>
</tr>
<tr>
<td>11</td>
<td>3.658195E-17</td>
<td>1.386745E-11</td>
<td>4.085088E+01</td>
<td>8.734127E+03</td>
</tr>
<tr>
<td>12</td>
<td>3.013838E+02</td>
<td>4.352953E+00</td>
<td>5.141852E-13</td>
<td>6.235738E+03</td>
</tr>
</tbody>
</table>

### DYNAMIC ANALYSIS FOR SEISMIC LOADS

--- PAGE NO. 5 ---

### MASS PARTICIPATION FACTORS IN PERCENT

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>85.32</td>
<td>85.325</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>98.01</td>
<td>0.00</td>
<td>0.00</td>
<td>98.014</td>
<td>0.00</td>
<td>85.325</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>98.01</td>
<td>0.00</td>
<td>85.325</td>
<td>85.325</td>
</tr>
</tbody>
</table>

--- PAGE NO. 6 ---

### DYNAMIC ANALYSIS FOR SEISMIC LOADS

--- PAGE NO. 5 ---

### MASS PARTICIPATION FACTORS IN PERCENT

<table>
<thead>
<tr>
<th>MODE</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>SUMM-X</th>
<th>SUMM-Y</th>
<th>SUMM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>85.32</td>
<td>85.325</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>98.01</td>
<td>0.00</td>
<td>0.00</td>
<td>98.014</td>
<td>0.00</td>
<td>85.325</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>98.01</td>
<td>0.00</td>
<td>85.325</td>
<td>85.325</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. British Design Examples

<table>
<thead>
<tr>
<th>Mode</th>
<th>Actual Modal Damping Used in Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.05000000</td>
</tr>
<tr>
<td>2</td>
<td>0.05000000</td>
</tr>
<tr>
<td>3</td>
<td>0.05000000</td>
</tr>
<tr>
<td>4</td>
<td>0.05000000</td>
</tr>
<tr>
<td>5</td>
<td>0.05000000</td>
</tr>
<tr>
<td>6</td>
<td>0.05000000</td>
</tr>
<tr>
<td>7</td>
<td>0.05000000</td>
</tr>
<tr>
<td>8</td>
<td>0.05000000</td>
</tr>
<tr>
<td>9</td>
<td>0.05000000</td>
</tr>
<tr>
<td>10</td>
<td>0.05000000</td>
</tr>
<tr>
<td>11</td>
<td>0.05000000</td>
</tr>
<tr>
<td>12</td>
<td>0.05000000</td>
</tr>
<tr>
<td>13</td>
<td>0.05000000</td>
</tr>
<tr>
<td>14</td>
<td>0.05000000</td>
</tr>
<tr>
<td>15</td>
<td>0.05000000</td>
</tr>
<tr>
<td>16</td>
<td>0.05000000</td>
</tr>
<tr>
<td>17</td>
<td>0.05000000</td>
</tr>
<tr>
<td>18</td>
<td>0.05000000</td>
</tr>
<tr>
<td>19</td>
<td>0.05000000</td>
</tr>
<tr>
<td>20</td>
<td>0.05000000</td>
</tr>
<tr>
<td>21</td>
<td>0.05000000</td>
</tr>
<tr>
<td>22</td>
<td>0.05000000</td>
</tr>
<tr>
<td>23</td>
<td>0.05000000</td>
</tr>
<tr>
<td>24</td>
<td>0.05000000</td>
</tr>
<tr>
<td>25</td>
<td>0.05000000</td>
</tr>
<tr>
<td>26</td>
<td>0.05000000</td>
</tr>
<tr>
<td>27</td>
<td>0.05000000</td>
</tr>
<tr>
<td>28</td>
<td>0.05000000</td>
</tr>
<tr>
<td>29</td>
<td>0.05000000</td>
</tr>
<tr>
<td>30</td>
<td>0.05000000</td>
</tr>
</tbody>
</table>

**DYNAMIC ANALYSIS FOR SEISMIC LOADS**

---

**-- PAGE NO. 7**
**Dynamic Analysis for Seismic Loads**

**Structure Type = Space**

<table>
<thead>
<tr>
<th>Joint</th>
<th>Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0015</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0173</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0157</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0190</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.0290</td>
<td>-0.0012</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>-0.4146</td>
<td>-0.0050</td>
<td>-0.0004</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0004</td>
</tr>
<tr>
<td>3</td>
<td>5.7319</td>
<td>0.0046</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0142</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>5.2883</td>
<td>-0.0015</td>
<td>-0.0005</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>-0.0138</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-6.1755</td>
<td>-0.0108</td>
<td>-0.0005</td>
<td>-0.0001</td>
<td>-0.0000</td>
<td>0.0145</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-0.0249</td>
<td>-0.0016</td>
<td>-0.0002</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>-0.3581</td>
<td>-0.0075</td>
<td>-0.0122</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0001</td>
<td>-0.0012</td>
</tr>
<tr>
<td>3</td>
<td>7.9529</td>
<td>0.0070</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0101</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>7.5699</td>
<td>-0.0021</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0113</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-8.3358</td>
<td>-0.0161</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>-0.0001</td>
<td>0.0088</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-0.0830</td>
<td>-0.0016</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0001</td>
</tr>
<tr>
<td>2</td>
<td>-0.0451</td>
<td>-0.0076</td>
<td>0.0004</td>
<td>0.0002</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0020</td>
</tr>
<tr>
<td>3</td>
<td>9.3683</td>
<td>0.0023</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0083</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>9.3201</td>
<td>-0.0069</td>
<td>0.0004</td>
<td>0.0002</td>
<td>-0.0000</td>
<td>-0.0104</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-9.4164</td>
<td>-0.0115</td>
<td>0.0004</td>
<td>0.0002</td>
<td>-0.0000</td>
<td>0.0062</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0014</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0174</td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0189</td>
</tr>
<tr>
<td>12</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0159</td>
</tr>
<tr>
<td>6</td>
<td>0.0261</td>
<td>-0.0012</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.3723</td>
<td>-0.0050</td>
<td>-0.0004</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>-0.0002</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>5.7484</td>
<td>-0.0046</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0142</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>6.1468</td>
<td>-0.0108</td>
<td>-0.0005</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>-0.0144</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-5.3500</td>
<td>-0.0015</td>
<td>-0.0005</td>
<td>-0.0001</td>
<td>0.0000</td>
<td>0.0139</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.0286</td>
<td>-0.0016</td>
<td>-0.0002</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.2940</td>
<td>-0.0075</td>
<td>-0.0122</td>
<td>-0.0000</td>
<td>0.0001</td>
<td>0.0003</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>7.9667</td>
<td>-0.0070</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0100</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>8.2813</td>
<td>-0.0161</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>0.0001</td>
<td>-0.0086</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-7.6521</td>
<td>-0.0214</td>
<td>-0.0124</td>
<td>-0.0000</td>
<td>0.0001</td>
<td>0.0114</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-0.0827</td>
<td>0.0016</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-0.3980</td>
<td>-0.0077</td>
<td>0.0004</td>
<td>0.0002</td>
<td>0.0000</td>
<td>0.0021</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9.3696</td>
<td>-0.0026</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0082</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>9.3271</td>
<td>-0.0119</td>
<td>0.0004</td>
<td>0.0002</td>
<td>0.0000</td>
<td>-0.0060</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-9.4121</td>
<td>-0.0068</td>
<td>0.0004</td>
<td>0.0002</td>
<td>0.0000</td>
<td>0.0105</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0001</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. British Design Examples

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>-0.4146</td>
<td>-0.0050</td>
<td>0.0004</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0004</td>
</tr>
<tr>
<td>11</td>
<td>5.7319</td>
<td>0.0046</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0142</td>
</tr>
<tr>
<td>12</td>
<td>-6.1755</td>
<td>-0.0108</td>
<td>0.0005</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0145</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.3723</td>
<td>-0.0050</td>
<td>0.0004</td>
<td>0.0001</td>
<td>-0.0000</td>
<td>-0.0002</td>
</tr>
<tr>
<td>11</td>
<td>7.9529</td>
<td>0.0070</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>-0.0145</td>
</tr>
<tr>
<td>12</td>
<td>-8.3358</td>
<td>-0.0161</td>
<td>0.0124</td>
<td>0.0001</td>
<td>0.0000</td>
<td>0.0088</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>0.0261</td>
<td>-0.0012</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.3723</td>
<td>-0.0050</td>
<td>0.0004</td>
<td>0.0001</td>
<td>-0.0000</td>
<td>-0.0142</td>
</tr>
<tr>
<td>3</td>
<td>5.7484</td>
<td>-0.0046</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0083</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
<td>0.0206</td>
<td>-0.0016</td>
<td>0.0002</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.2940</td>
<td>-0.0076</td>
<td>0.0122</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0013</td>
</tr>
<tr>
<td>3</td>
<td>7.9667</td>
<td>-0.0070</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0100</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>0.0206</td>
<td>-0.0016</td>
<td>0.0002</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>0.2940</td>
<td>-0.0076</td>
<td>0.0122</td>
<td>0.0000</td>
<td>-0.0001</td>
<td>0.0013</td>
</tr>
<tr>
<td>3</td>
<td>7.9667</td>
<td>-0.0070</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>-0.0000</td>
<td>-0.0100</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>-0.0027</td>
<td>-0.0016</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>18</td>
<td>-0.0027</td>
<td>-0.0016</td>
<td>-0.0000</td>
<td>-0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joint Load</th>
<th>X-Trans</th>
<th>Y-Trans</th>
<th>Z-Trans</th>
<th>X-RotAN</th>
<th>Y-RotAN</th>
<th>Z-RotAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. British Design Examples

#### Dynamic Analysis for Seismic Loads

**Support Reactions - Unit Kgs Meters**  
**Structure Type = Space**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>61.39</td>
<td>1009.30</td>
<td>0.98</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>872.77</td>
<td>3562.50</td>
<td>-19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-2356.64</td>
<td>-3311.14</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>-1422.49</td>
<td>1260.65</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>12</td>
<td>3290.79</td>
<td>7882.94</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>1</td>
<td>61.39</td>
<td>1009.30</td>
<td>-0.98</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-872.77</td>
<td>3562.50</td>
<td>19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>-2392.50</td>
<td>3308.29</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>11</td>
<td>-3326.65</td>
<td>7880.23</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>12</td>
<td>1458.35</td>
<td>1263.66</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

#### Dynamic Analysis for Seismic Loads

**Member End Forces - Structure Type = Space**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1009.30</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>-666.89</td>
<td>61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>-3.43</td>
<td>-214.85</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>3562.50</td>
<td>872.77</td>
<td>-19.74</td>
<td>0.00</td>
<td>69.09</td>
<td>-3054.69</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-3562.50</td>
<td>872.77</td>
<td>19.74</td>
<td>0.00</td>
<td>65.66</td>
<td>8248.24</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-3311.14</td>
<td>2356.64</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3311.14</td>
<td>2356.64</td>
<td>0.00</td>
<td>0.00</td>
<td>8248.24</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>1260.65</td>
<td>1422.49</td>
<td>-18.76</td>
<td>0.00</td>
<td>65.66</td>
<td>4978.71</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>7882.94</td>
<td>-3290.79</td>
<td>-18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-7548.53</td>
<td>3290.79</td>
<td>18.76</td>
<td>0.00</td>
<td>65.66</td>
<td>-11517.77</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**STAAD.Pro 4900 User Manual**
### Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Moment-Y</th>
<th>Moment-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>2</td>
<td>3</td>
<td>-464.90</td>
<td>61.39</td>
<td>-5.97</td>
<td>-0.01</td>
<td>-1.72</td>
<td>-325.34</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>3</td>
<td>3562.50</td>
<td>-872.77</td>
<td>112.31</td>
<td>0.24</td>
<td>-79.35</td>
<td>3054.69</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>-464.90</td>
<td>61.39</td>
<td>-5.97</td>
<td>-0.01</td>
<td>-1.72</td>
<td>-325.34</td>
<td>-325.34</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>-3562.50</td>
<td>872.77</td>
<td>-112.31</td>
<td>0.24</td>
<td>-8248.24</td>
<td>-8248.24</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>3562.50</td>
<td>-872.77</td>
<td>112.31</td>
<td>0.24</td>
<td>-8248.24</td>
<td>3054.69</td>
<td></td>
</tr>
</tbody>
</table>

**Dynamic Analysis for Seismic Loads**

---

**Page No. 13**

---

**MEMBER END FORCES**, **STRUCTURE TYPE = SPACE**

---

**ALL UNITS ARE -- KGS METE (LOCAL)**

---

**STAAD.Pro**

---

**User Manual**
### Application Examples
#### EX. British Design Examples

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>11</td>
<td>41.65</td>
<td>2832.69</td>
<td>-111.63</td>
<td>0.35</td>
<td>-122.13</td>
<td>4570.39</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>11</td>
<td>6886.37</td>
<td>8900.34</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-12336.68</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>11</td>
<td>7119.69</td>
<td>5881.00</td>
<td>-176.60</td>
<td>0.36</td>
<td>-123.83</td>
<td>7456.29</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>11</td>
<td>6653.85</td>
<td>11919.69</td>
<td>-176.60</td>
<td>0.36</td>
<td>-123.83</td>
<td>17217.07</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>1009.45</td>
<td>61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>-667.04</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>3.43</td>
<td>214.85</td>
<td></td>
</tr>
</tbody>
</table>

---

### Dynamic Analysis for Seismic Loads

- **MEMBER END FORCES**
- **STRUCTURE TYPE = SPACE**

---

All units are -- KGS METE (LOCAL)

---

### Application Examples
#### EX. British Design Examples

<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1</td>
<td>13</td>
<td>1076.15</td>
<td>61.39</td>
<td>-0.98</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>-667.04</td>
<td>-61.39</td>
<td>0.98</td>
<td>0.00</td>
<td>3.43</td>
<td>214.85</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>13</td>
<td>3562.50</td>
<td>872.77</td>
<td>19.74</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>3562.50</td>
<td>872.77</td>
<td>-19.74</td>
<td>0.00</td>
<td>-69.07</td>
<td>3054.69</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>13</td>
<td>3308.29</td>
<td>2392.50</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>3308.29</td>
<td>2392.50</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>8373.75</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>13</td>
<td>7510.81</td>
<td>3226.65</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>7510.81</td>
<td>3226.65</td>
<td>18.76</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>13</td>
<td>1263.66</td>
<td>1458.35</td>
<td>18.76</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>14</td>
<td>641.15</td>
<td>61.39</td>
<td>-5.97</td>
<td>0.81</td>
<td>9.03</td>
<td>214.85</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>641.15</td>
<td>61.39</td>
<td>5.97</td>
<td>0.81</td>
<td>1.72</td>
<td>325.34</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>14</td>
<td>3562.50</td>
<td>872.77</td>
<td>-122.28</td>
<td>0.22</td>
<td>79.33</td>
<td>3054.69</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>3562.50</td>
<td>872.77</td>
<td>122.28</td>
<td>-0.22</td>
<td>122.76</td>
<td>4625.67</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>14</td>
<td>3308.16</td>
<td>2288.65</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-8373.75</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>3308.16</td>
<td>2288.65</td>
<td>-0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>12493.31</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>14</td>
<td>7511.81</td>
<td>3222.80</td>
<td>-118.25</td>
<td>0.23</td>
<td>88.36</td>
<td>-11643.29</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>7511.81</td>
<td>3222.80</td>
<td>118.25</td>
<td>-0.23</td>
<td>124.49</td>
<td>17444.32</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>14</td>
<td>895.48</td>
<td>1354.49</td>
<td>-118.25</td>
<td>0.23</td>
<td>88.36</td>
<td>5104.22</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>895.48</td>
<td>1354.49</td>
<td>118.25</td>
<td>-0.23</td>
<td>124.49</td>
<td>-7542.31</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>15</td>
<td>206.02</td>
<td>-182.87</td>
<td>-5.96</td>
<td>-0.01</td>
<td>-1.71</td>
<td>-308.77</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-39.71</td>
<td>182.87</td>
<td>5.96</td>
<td>0.01</td>
<td>11.84</td>
<td>-2.12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>15</td>
<td>257.08</td>
<td>-2789.79</td>
<td>-111.63</td>
<td>-0.33</td>
<td>-122.14</td>
<td>-4566.17</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>257.08</td>
<td>2789.79</td>
<td>111.63</td>
<td>0.33</td>
<td>311.91</td>
<td>-176.47</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>15</td>
<td>-6533.99</td>
<td>9068.05</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-0.00</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>-6533.99</td>
<td>9068.05</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-2889.65</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>15</td>
<td>-6070.89</td>
<td>-12040.72</td>
<td>-117.59</td>
<td>-0.34</td>
<td>-123.84</td>
<td>-17401.14</td>
<td></td>
</tr>
</tbody>
</table>
## Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Moment-Y</th>
<th>Moment-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>4</td>
<td>9442.80</td>
<td>6886.14</td>
<td>-0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>2793.95</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-9442.80</td>
<td>-6886.14</td>
<td>0.00</td>
<td>-0.00</td>
<td>0.00</td>
<td>9601.10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>4</td>
<td>6423.46</td>
<td>6364.76</td>
<td>-0.70</td>
<td>0.09</td>
<td>1.54</td>
<td>8367.98</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-6423.46</td>
<td>-5506.82</td>
<td>0.70</td>
<td>-0.09</td>
<td>1.57</td>
<td>3046.45</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>12462.15</td>
<td>-7407.52</td>
<td>-0.70</td>
<td>0.69</td>
<td>-0.29</td>
<td>-245.19</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>12462.15</td>
<td>8265.46</td>
<td>0.70</td>
<td>-0.09</td>
<td>1.57</td>
<td>-10834.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>178</td>
<td>61.39</td>
<td>129.40</td>
<td>-0.00</td>
<td>0.03</td>
<td>-141.09</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>178</td>
<td>-61.39</td>
<td>-19.82</td>
<td>0.00</td>
<td>0.03</td>
<td>141.09</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>17</td>
<td>872.77</td>
<td>2212.50</td>
<td>-0.01</td>
<td>1.76</td>
<td>0.27</td>
<td>1193.12</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-872.77</td>
<td>-675.00</td>
<td>0.01</td>
<td>1.76</td>
<td>0.24</td>
<td>2343.13</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>17</td>
<td>-6423.46</td>
<td>6364.76</td>
<td>-0.70</td>
<td>0.09</td>
<td>-0.30</td>
<td>-245.19</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>12462.15</td>
<td>-7407.52</td>
<td>-0.70</td>
<td>0.70</td>
<td>-0.30</td>
<td>-245.19</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>12462.15</td>
<td>8265.46</td>
<td>0.70</td>
<td>-0.09</td>
<td>1.57</td>
<td>-10834.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>178</td>
<td>61.39</td>
<td>129.40</td>
<td>-0.00</td>
<td>0.03</td>
<td>-141.09</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>178</td>
<td>-61.39</td>
<td>-19.82</td>
<td>0.00</td>
<td>0.03</td>
<td>141.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>17</td>
<td>872.77</td>
<td>2212.50</td>
<td>-0.01</td>
<td>1.76</td>
<td>0.27</td>
<td>1193.12</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>-872.77</td>
<td>-675.00</td>
<td>0.01</td>
<td>1.76</td>
<td>0.24</td>
<td>2343.13</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>17</td>
<td>-6423.46</td>
<td>6364.76</td>
<td>-0.70</td>
<td>0.09</td>
<td>-0.30</td>
<td>-245.19</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>12462.15</td>
<td>-7407.52</td>
<td>-0.70</td>
<td>0.70</td>
<td>-0.30</td>
<td>-245.19</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>12462.15</td>
<td>8265.46</td>
<td>0.70</td>
<td>-0.09</td>
<td>1.57</td>
<td>-10834.23</td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>178</td>
<td>61.39</td>
<td>129.40</td>
<td>-0.00</td>
<td>0.03</td>
<td>-141.09</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>178</td>
<td>-61.39</td>
<td>-19.82</td>
<td>0.00</td>
<td>0.03</td>
<td>141.09</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Dynamic Analysis for Seismic Loads**

**Member End Forces**  
**Structure Type = Space**

---

**All Units Are -- KGS METE (LOCAL)**
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>LOAD</th>
<th>JT</th>
<th>AXIAL</th>
<th>SHEAR-Y</th>
<th>SHEAR-Z</th>
<th>TORSION</th>
<th>MOM-Y</th>
<th>MOM-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12</td>
<td>6423.46</td>
<td>6364.76</td>
<td>0.70</td>
<td>-0.09</td>
<td>0.30</td>
<td>2541.45</td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>-6423.46</td>
<td>-5506.82</td>
<td>-0.70</td>
<td>0.09</td>
<td>-1.57</td>
<td>8367.98</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>12</td>
<td>-12462.15</td>
<td>-7407.52</td>
<td>0.70</td>
<td>-0.09</td>
<td>0.30</td>
<td>-10834.23</td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>12462.15</td>
<td>8265.46</td>
<td>-0.70</td>
<td>0.09</td>
<td>-1.57</td>
<td>8367.98</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>61.39</td>
<td>129.40</td>
<td>0.00</td>
<td>0.03</td>
<td>-0.01</td>
<td>67.81</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-61.39</td>
<td>-19.82</td>
<td>0.00</td>
<td>0.03</td>
<td>-0.01</td>
<td>67.81</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>872.77</td>
<td>2212.50</td>
<td>0.01</td>
<td>1.76</td>
<td>-0.27</td>
<td>1193.12</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-872.77</td>
<td>-675.00</td>
<td>0.01</td>
<td>1.76</td>
<td>-0.27</td>
<td>1193.12</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>20</td>
<td>-725.43</td>
<td>3432.33</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>725.43</td>
<td>-3432.33</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>20</td>
<td>208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>11266.44</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>11266.44</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>11266.44</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>11266.44</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>61.39</td>
<td>-6.07</td>
<td>0.00</td>
<td>-0.03</td>
<td>-0.01</td>
<td>141.09</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-61.39</td>
<td>123.46</td>
<td>0.00</td>
<td>0.03</td>
<td>0.01</td>
<td>-53.20</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>21</td>
<td>872.77</td>
<td>-450.00</td>
<td>0.01</td>
<td>-1.64</td>
<td>0.23</td>
<td>-2343.13</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-872.77</td>
<td>2373.51</td>
<td>0.01</td>
<td>1.64</td>
<td>-0.26</td>
<td>-975.62</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>21</td>
<td>667.43</td>
<td>3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-667.43</td>
<td>-3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-9929.10</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>20</td>
<td>208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>-8744.57</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>-8744.57</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>1659.58</td>
<td>8265.46</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>2879.93</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-1659.58</td>
<td>-8265.46</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>2879.93</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>61.39</td>
<td>-6.07</td>
<td>0.00</td>
<td>-0.03</td>
<td>-0.01</td>
<td>141.09</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-61.39</td>
<td>123.46</td>
<td>0.00</td>
<td>0.03</td>
<td>0.01</td>
<td>-53.20</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>21</td>
<td>872.77</td>
<td>-450.00</td>
<td>0.01</td>
<td>-1.64</td>
<td>0.23</td>
<td>-2343.13</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-872.77</td>
<td>2373.51</td>
<td>0.01</td>
<td>1.64</td>
<td>-0.26</td>
<td>-975.62</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>21</td>
<td>667.43</td>
<td>3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-667.43</td>
<td>-3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-9929.10</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>20</td>
<td>208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>-8744.57</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>-8744.57</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>1659.58</td>
<td>8265.46</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>2879.93</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-1659.58</td>
<td>-8265.46</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>2879.93</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>61.39</td>
<td>-6.07</td>
<td>0.00</td>
<td>-0.03</td>
<td>-0.01</td>
<td>141.09</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-61.39</td>
<td>123.46</td>
<td>0.00</td>
<td>0.03</td>
<td>0.01</td>
<td>-53.20</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>21</td>
<td>872.77</td>
<td>-450.00</td>
<td>0.01</td>
<td>-1.64</td>
<td>0.23</td>
<td>-2343.13</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-872.77</td>
<td>2373.51</td>
<td>0.01</td>
<td>1.64</td>
<td>-0.26</td>
<td>-975.62</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>21</td>
<td>667.43</td>
<td>3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>395.71</td>
<td></td>
</tr>
<tr>
<td></td>
<td>22</td>
<td>-667.43</td>
<td>-3441.37</td>
<td>0.00</td>
<td>0.00</td>
<td>-0.00</td>
<td>-9929.10</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>20</td>
<td>208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>-8744.57</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-208.72</td>
<td>208.72</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>-8744.57</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>20</td>
<td>1659.58</td>
<td>8265.46</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>2879.93</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>-1659.58</td>
<td>-8265.46</td>
<td>0.01</td>
<td>1.79</td>
<td>-0.28</td>
<td>2879.93</td>
<td></td>
</tr>
</tbody>
</table>

**DYNAMIC ANALYSIS FOR SEISMIC LOADS**

**MEMBER END FORCES, STRUCTURE TYPE = SPACE**

**ALL UNITS ARE -- KGS  METE (LOCAL )**
<table>
<thead>
<tr>
<th>Member</th>
<th>Load</th>
<th>JT</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Mom-Y</th>
<th>Mom-Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>26</td>
<td>1</td>
<td>7</td>
<td>354.75</td>
<td>30.17</td>
<td>-0.01</td>
<td>0.00</td>
<td>-0.02</td>
<td>16.58</td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>-384.72</td>
<td>25.75</td>
<td>0.81</td>
<td>-0.00</td>
<td>0.05</td>
<td>-10.94</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>4933.53</td>
<td>21.16</td>
<td>-0.65</td>
<td>0.10</td>
<td>-0.83</td>
<td>59.49</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>-4933.53</td>
<td>21.16</td>
<td>0.65</td>
<td>-0.10</td>
<td>2.48</td>
<td>-5.54</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>7</td>
<td>14950.87</td>
<td>171.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>31.13</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>-14950.87</td>
<td>171.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-403.31</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>7</td>
<td>20238.35</td>
<td>119.78</td>
<td>-0.66</td>
<td>0.10</td>
<td>-0.85</td>
<td>42.93</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>-20188.32</td>
<td>175.70</td>
<td>0.66</td>
<td>-0.10</td>
<td>2.53</td>
<td>-419.79</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>7</td>
<td>-9661.80</td>
<td>222.44</td>
<td>-0.66</td>
<td>0.10</td>
<td>-0.85</td>
<td>109.20</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>9711.83</td>
<td>166.53</td>
<td>0.66</td>
<td>-0.10</td>
<td>2.53</td>
<td>386.83</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>1</td>
<td>11</td>
<td>367.93</td>
<td>28.33</td>
<td>-0.01</td>
<td>0.00</td>
<td>-0.02</td>
<td>15.34</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>-317.90</td>
<td>24.65</td>
<td>0.01</td>
<td>-0.00</td>
<td>0.05</td>
<td>-10.79</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>11</td>
<td>5111.42</td>
<td>15.45</td>
<td>0.68</td>
<td>-0.10</td>
<td>55.28</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>-5111.42</td>
<td>15.45</td>
<td>-0.68</td>
<td>0.10</td>
<td>-2.59</td>
<td>-17.03</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>367.93</td>
<td>28.33</td>
<td>0.01</td>
<td>-0.00</td>
<td>0.02</td>
<td>15.34</td>
<td></td>
</tr>
<tr>
<td></td>
<td>17</td>
<td>-317.90</td>
<td>24.65</td>
<td>0.01</td>
<td>-0.00</td>
<td>0.05</td>
<td>-10.79</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>7</td>
<td>14950.87</td>
<td>171.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>31.13</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>-14950.87</td>
<td>171.11</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-403.31</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>7</td>
<td>20238.35</td>
<td>119.78</td>
<td>-0.66</td>
<td>0.10</td>
<td>-0.85</td>
<td>42.93</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>-20188.32</td>
<td>175.70</td>
<td>0.66</td>
<td>-0.10</td>
<td>2.53</td>
<td>-419.79</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>7</td>
<td>-9661.80</td>
<td>222.44</td>
<td>-0.66</td>
<td>0.10</td>
<td>-0.85</td>
<td>109.20</td>
<td></td>
</tr>
<tr>
<td></td>
<td>19</td>
<td>9711.83</td>
<td>166.53</td>
<td>0.66</td>
<td>-0.10</td>
<td>2.53</td>
<td>386.83</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>1</td>
<td>15</td>
<td>354.75</td>
<td>30.17</td>
<td>0.01</td>
<td>-0.00</td>
<td>0.02</td>
<td>16.58</td>
</tr>
</tbody>
</table>

Dynamic Analysis for Seismic Loads

Member End Forces

Structure Type: Space

All Units Are: Kgs Meters (Local)
<table>
<thead>
<tr>
<th>MEMBER</th>
<th>TABLE</th>
<th>RESULT/LOCATION</th>
<th>CRITICAL COND/</th>
<th>RATIO/LOADING/</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ST</td>
<td>W310X97</td>
<td>321</td>
<td>12</td>
<td></td>
</tr>
<tr>
<td>73.95</td>
<td>-0.64</td>
<td>112.95</td>
<td>3.50</td>
<td></td>
</tr>
<tr>
<td>73.92</td>
<td>-0.64</td>
<td>-114.18</td>
<td>3.50</td>
<td></td>
</tr>
<tr>
<td>71.97</td>
<td>1.22</td>
<td>169.22</td>
<td>1.80</td>
<td></td>
</tr>
<tr>
<td>65.24</td>
<td>1.21</td>
<td>168.84</td>
<td>0.00</td>
<td></td>
</tr>
<tr>
<td>73.92</td>
<td>-0.64</td>
<td>-114.18</td>
<td>3.50</td>
<td></td>
</tr>
<tr>
<td>71.94</td>
<td>1.22</td>
<td>-171.07</td>
<td>1.80</td>
<td></td>
</tr>
<tr>
<td>59.54</td>
<td>1.21</td>
<td>-170.65</td>
<td>0.00</td>
<td></td>
</tr>
</tbody>
</table>

DYNAMIC ANALYSIS FOR SEISMIC LOADS

STEEL DESIGN

STAAD.PRO CODE CHECKING - S16-14 (v1.0)

ALL UNITS ARE - KN MET (UNLESS OTHERWISE NOTED)
### Application Examples

**EX. British Design Examples**

<table>
<thead>
<tr>
<th>Member</th>
<th>Section</th>
<th>Design Case</th>
<th>Load Case</th>
<th>Location</th>
<th>Critical Cond</th>
<th>Ratio</th>
<th>Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 ST</td>
<td>W310x97</td>
<td>Cl. 13.8.3</td>
<td>0.321</td>
<td>12</td>
<td>73.95 C</td>
<td>0.64</td>
<td>112.95</td>
</tr>
<tr>
<td>9 ST</td>
<td>W310x97</td>
<td>Cl. 13.8.3</td>
<td>0.467</td>
<td>12</td>
<td>71.97 C</td>
<td>-1.22</td>
<td>169.22</td>
</tr>
<tr>
<td>10 ST</td>
<td>W310x97</td>
<td>Cl. 13.8.3</td>
<td>0.324</td>
<td>11</td>
<td>73.92 C</td>
<td>0.64</td>
<td>-114.18</td>
</tr>
<tr>
<td>11 ST</td>
<td>W310x97</td>
<td>Cl. 13.8.3</td>
<td>0.471</td>
<td>11</td>
<td>71.94 C</td>
<td>-1.22</td>
<td>-171.07</td>
</tr>
<tr>
<td>12 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.798</td>
<td>12</td>
<td>16.27 C</td>
<td>0.00</td>
<td>110.49</td>
</tr>
<tr>
<td>13 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.776</td>
<td>11</td>
<td>15.71 C</td>
<td>0.00</td>
<td>107.46</td>
</tr>
<tr>
<td>14 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.864</td>
<td>12</td>
<td>122.21 T</td>
<td>-0.02</td>
<td>106.25</td>
</tr>
<tr>
<td>15 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.798</td>
<td>12</td>
<td>16.27 C</td>
<td>0.00</td>
<td>110.49</td>
</tr>
<tr>
<td>16 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.776</td>
<td>11</td>
<td>15.71 C</td>
<td>0.00</td>
<td>107.46</td>
</tr>
<tr>
<td>17 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.844</td>
<td>11</td>
<td>123.57 T</td>
<td>-0.02</td>
<td>103.34</td>
</tr>
<tr>
<td>18 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.798</td>
<td>12</td>
<td>16.27 C</td>
<td>-0.00</td>
<td>110.49</td>
</tr>
<tr>
<td>19 ST</td>
<td>W250x39</td>
<td>Cl. 13.8</td>
<td>0.776</td>
<td>11</td>
<td>15.71 C</td>
<td>-0.00</td>
<td>107.46</td>
</tr>
</tbody>
</table>

**ALL UNITS ARE - KN MET (UNLESS OTHERWISE NOTED)**

---

**STAAD.Pro Code Checking - S16-14 (v1.0)**

---

**STAAD.Pro User Manual**
<table>
<thead>
<tr>
<th>Member</th>
<th>Section</th>
<th>Pass</th>
<th>Cl.</th>
<th>Value</th>
<th>Value</th>
<th>Value</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>ST</td>
<td>W250X39</td>
<td>13.9.1</td>
<td>0.844</td>
<td>11</td>
<td>123.57</td>
<td>103.34</td>
</tr>
<tr>
<td>21</td>
<td>ST</td>
<td>C200X17</td>
<td>13.9.1</td>
<td>0.009</td>
<td>12</td>
<td>1.34</td>
<td>0.00</td>
</tr>
<tr>
<td>22</td>
<td>ST</td>
<td>C200X17</td>
<td>13.8.3</td>
<td>0.118</td>
<td>11</td>
<td>1.16</td>
<td>-3.18</td>
</tr>
<tr>
<td>23</td>
<td>ST</td>
<td>C200X17</td>
<td>13.9.1</td>
<td>0.009</td>
<td>12</td>
<td>1.34</td>
<td>0.00</td>
</tr>
<tr>
<td>24</td>
<td>ST</td>
<td>C200X17</td>
<td>13.8.3</td>
<td>0.118</td>
<td>11</td>
<td>1.16</td>
<td>0.00</td>
</tr>
<tr>
<td>25</td>
<td>ST</td>
<td>L152X152X13</td>
<td>13.8.3</td>
<td>0.632</td>
<td>12</td>
<td>200.71</td>
<td>4.24</td>
</tr>
<tr>
<td>26</td>
<td>ST</td>
<td>L152X152X13</td>
<td>13.8.3</td>
<td>0.630</td>
<td>11</td>
<td>197.98</td>
<td>4.12</td>
</tr>
<tr>
<td>27</td>
<td>ST</td>
<td>L152X152X13</td>
<td>13.8.3</td>
<td>0.632</td>
<td>12</td>
<td>200.71</td>
<td>4.24</td>
</tr>
<tr>
<td>28</td>
<td>ST</td>
<td>L152X152X13</td>
<td>13.8.3</td>
<td>0.630</td>
<td>11</td>
<td>197.98</td>
<td>4.12</td>
</tr>
<tr>
<td>29</td>
<td>ST</td>
<td>C200X17</td>
<td>13.8.3</td>
<td>0.444</td>
<td>11</td>
<td>0.00</td>
<td>-11.19</td>
</tr>
</tbody>
</table>

***NOTE: OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL MEMBERS***

79. FINISH

******* END OF THE STAAD.Pro RUN *******
**** DATE= APR 14,2019 TIME= 22:54:26 ****

* For technical assistance on STAAD.Pro, please visit *
* http://www.bentley.com/en/support/ *
* Details about additional assistance from *
* Bentley and Partners can be found at program menu *
* Help->Technical Support *
* Copyright (c) 1997-2017 Bentley Systems, Inc. *
EX. Modeling Examples

These examples demonstrate using some of modeling features in STAAD.Pro.

EX. Meshed Wall-Slab Connection

This example demonstrates how to ensure finite element mesh compatibility between two adjoining meshed entities.

The Model Data

Use the Copy to clipboard button below to copy and paste this model data into an empty STAAD input file if you want to work through this example yourself.

STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 18 0 0; 3 30 0 9; 4 30 0 30; 5 6 0 30; 6 0 0 21; 7 6 0 9;
8 18 0 24; 9 6 20 9; 10 18 20 24;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 1; 7 7 8; 8 9 10; 9 7 9; 10 8 10;
FINISH

Related Links
• M. To define a slab/wall connection (on page 660)

EX. To create a new view

To create a new view that isolates the slab members for meshing, use the following procedure.

1. On the View ribbon tab, select the Front View tool in the Tools group.
The view rotates to this angle.

2. Click-and-drag the mouse pointer from the top-left towards the bottom-right of the slab members.

   **Note:** You may need to select the **Beam Cursor** tool on the **Select** ribbon tab first.

3. On the **View** ribbon tab, select the **New View** tool in the **Views** group.

   The **New View** dialog opens.

   ![New View dialog](image)

4. Select the **Display the view in the active window** option and then click **OK**.

   The new view is displayed in the active view window. Only the selected slab members are currently displayed. The wall members are still contained in the model. They are just hidden from this view.

5. On the **View** ribbon tab, select the **Isometric View** tool in the **Tools** group.

EX. To create a parametric model

To create a slab finite element mesh, use the following procedure.

1. Select the **Parametric Models** tool in the Plate group on the **Geometry** ribbon tab.
The **Parametric Models** dialog opens. The structure is displayed as a series of dashed lines to signify this mode can be used to experiment with various settings.

2. Click **Add** in the **Parametric Models** dialog.

   The **New Mesh Model** dialog opens.

3. Specify the parametric model parameters:
   a. (Optional) Type a **Name** for the slab mesh.
   b. Select **Slab** from the **Type** drop-down list.
   c. Set the **Use nodes and beams that occur on or inside the outer boundary as additional density lines and points?** option.

4. Click **OK**.

   The mouse pointer changes to the add surface cursor.

5. Click the corner nodes of the slab in a clockwise order, clicking on the first node selected to complete the boundary definition.

   ![Diagram of a slab model with nodes and beams]

   The **Mesh Parameters** dialog opens.

6. Specify the meshing parameters:
   Leave the **Element Type** as Triangular, the **Boundary method** as (C) Optimized, and the **Default Number of Divisions of Boundary Edges** as zero.
a. Select Basic from the Meshing Method drop-down list.
b. Type a Target element size of 100 (in).
c. Click OK.
A message dialog prompts if you want to add any openings.

7. Click No.
The preliminary mesh is displayed in the view window. The mesh parameters are displayed in the Parametric Models dialog.

8. In the Parametric Models dialog, select the Meshed Surface 1 [Slab] entry and then click Merge Mesh.
EX. To designate the wall-slab connection

To mesh the wall such that it is compatible with the slab mesh, use the following procedure.

1. In the Analytical Modeling workflow bar, select Geometry.
   The Parametric Modeling dialog closes and the plate model is displayed.
2. On the View ribbon tab, select the Display Whole Structure tool in the Tools group.

   ![Image of Parametric Modeling]

   The standard view is restored, displaying the hidden portions to the view.
3. (Optional) On the Select ribbon tab, select the All tool in the Beams group.

   ![Image of Select ribbon]

   Tip: It is not necessary to select the members in order to perform the next steps. However, this can serve as an easy way to identify the edges of the slab-wall interface when selecting the vertices.
4. On the Geometry ribbon tab, select the Wall/Slab Connection tool in the Structure group.

   ![Image of Geometry ribbon]
5. Starting with the two nodes at the base, select the four corner nodes of the wall in a clockwise order, clicking on the first selected node a second time to complete the boundary definition.

The Division along wall dialog opens.

Note: As the divisions along the base of the wall must be compatible with the slab density line, this value will control the meshing along the sides.

6. Click OK.
   The default value of 10 divisions is sufficient for this example.

7. On the Utilities ribbon tab, select the Plate Tools > Plate Connectivity tool in the Geometry Tools group.
   A message dialog opens indicating that no overlapping plates are found. This indicates also that the slab and wall are properly connected.
EX. Building Planner Workflow Example

This short tutorial will guide you through the creation of a small two-story concrete frame building using the Building Planner workflow.
EX. Create the model and plan details

1. On the Start page, select **New**. The New page opens to the **Model Info** tab.
2. Type **Building Example** in the **File Name** field and then select a **Location** to store the file.
3. Select the **Building** option for the **Type** of model.
4. Select **Metric** for the **Units**.

**Note:** You can use either system of units in the STAAD.Pro Analytical Workflow once you generate the analysis model. However, the Building Planner application only operates in metric units.

5. (Optional) Select the **Job Info** tab to add project member names, dates, project description data, etc. You can also associate your STAAD project with a ProjectWise Project here.
6. Click **Create**.

The STAAD.Pro window closes and the STAAD Building Planner application opens. The **Start** dialog opens.

7. Provide the project details:
   a. Type **Building Example** in the **Project** field.
   b. Type **Demo** in the **Client** field.
   c. Type your name or initials in the **Engineer** field.
   d. Type **2** in the **No. of Levels** field.
   e. Type **5 (m)** in the **Founding Depth** field.
This value is the distance from the top of foundations to the first floor.

8. Click **Create Project**.
   The **New Plan** dialog opens.

9. Enter in the plan details and assign it to the levels:
   a. Type 4 (m) in the **Height of Level Above** field.
   b. Click in the **Assign Levels** drop-down list and then click in the fields for both levels 1 and 2 such that **Plan1** is displayed there.
   This action assigns the typical plan to both levels.
   c. In the **Import Plan** group, select the **Create Plan Graphically** option.
   You will use the graphical interface to create a simple plan in this tutorial.
   Leave the remaining options in the dialog their default values.
10. Click Create Plan.
   The Slab Details (Rectangle) dialog opens.

   Proceed to the next set of steps to generate the slab.

**EX. Create the slab layout**

This procedure continues from the previous steps in which you created the model and plan details in building planner.

1. In the Slab Details (Rectangle) dialog, type 7 (m) in the Breadth (B) field.
   This updates the slab loading direction diagram in the dialog. Leave all other fields and options their defaults.
2. Click **OK**.
   A single slab section is added to the graphical view.

3. Select **Slab > Create Slab Rectangle**.

4. Click inside the existing slab near the top, left corner.
   A rectangular window is now pinned to the nearest corner and "rubber banded" to your mouse pointer.

5. Click at a point near where you want the opposite corner of the slab section placed.
   This should roughly be in line with the existing slab edge.
   The **Slab Details (Rectangle)** dialog opens. The same parameters from your first slab are entered.

6. Either:
   - click **OK**
   - or
   - press **<Enter>**
   The new slab section is added adjacent to the existing slab edge.

7. Repeat steps 5 and 6 to complete the 2 bay by 3 bay slab area.
Tip: You do not need to reselect the tool from the menu as it remains active after you create a slab.

Click the Save tool to save your progress so far.

**EX. Add the beams and columns**

1. Select Beam > Auto Beam.
   The Auto Beam Detail dialog opens.
2. Customize the beam properties to use:
   a. Type 0.30 (m) for the Internal Beam Breadth.
   b. Type 0.25 (m) for the External Beam Breadth.
   Leave the remaining parameters their default values.
3. Click **OK**.
   The beams are automatically added around the slab panel edges.
4. If the Select Beam to Break dialog opens, break the beams spanning 7 m.
   That is, such that the girder spanning horizontally on the screen is continuous.
5. Repeat step 4 if necessary.
6. Select **Column > Auto Column**.
   Square columns are automatically added at each beam intersection.

EX. Check for errors and align columns

1. Perform error checks:
   a. Select **Analysis > Slab Error Check**.
      This should result in no warnings or messages.
   b. Select **Analysis > Beam Error Check**.
      This should result in no warnings or messages.
2. Select **Analysis > Finalize Plan (with Continuity)**.
   A message dialog confirms the plan is finalized.
3. Select **Workspace > 3D Frame**.
   If the **3D View** window opens, close this window.
   **Note:** You are now in the frame workspace, which has different menu options.
4. Select **Assign > Column Sizing and Orientation**.
   The **Column Sizing** window opens.
5. Set the column offset so that the faces of the columns align with the faces of the exterior beams:
   a. Click Column C1 to select it (the top, left column).
   b. In the Column Offset And Beta Angle diagram, click corner 4 (the top, left corner).
   c. Click **Update Offset Data & Beta Angle**.  
      The column face is aligned to this point on the beams.

6. Repeat step 6 for each of the edge and corner columns, selecting the corresponding edge or corner for alignment.
7. Close the Column Sizing window.

EX. Assign lateral load data and generate the analysis model

1. Select Assign > Wind Parameters.
   The Wind Parameters dialog opens. Here you can select the appropriate Indian or American code for wind loads and assign parameters. For this example, you will accept the default parameters.
2. Select IS 875-1987 from the **Design Code** drop-down and then click OK.

3. Select **Assign > Seismic Parameters**.

   The **Seismic Parameters** dialog opens. Here you can select the appropriate Indian or American code for wind loads and assign parameters. For this example, you will accept the default parameters.
4. Select **Analysis > Generate Analysis File**. The **Space Frame Generation** dialog opens.

5. Click **Generate**. A warning dialog asks to overwrite the existing file, click **Yes**.

The STAAD Building Planner window closes and the analysis model opens in STAAD.Pro. All gravity loads, lateral loads, load combinations, and analysis commands have been added to the STAAD.Pro model. You can run the analysis in the Analytical Modeling workflow with no further actions necessary.

**EX. Steel Design Examples**

**EX. Connection Design Example**

This short tutorial presents an overview of design and detailing several steel connections. This example uses US Example no. 7, which is included with the installation of STAAD.Pro.
To start the Connection Design workflow

To open the example file in the Connection Design workflow, use the following procedure.

Using the Connection Design workflow requires a RAM Connection license. This license can incur additional cost to your organization.

1. a. Close any open model files in STAAD.Pro.
   b. On the Start page, check **RAM Connection** in the **Additional Licenses** panel.
      The **Additional License Selected** message opens to indicate that this can incur additional cost.
   c. Click Yes to activate this license for STAAD.Pro.
2. Open the file example file US.7. Modeling Offset Connections in a Frame.std in the **Analytical Modeling** workflow.
   This file is typically found in the
   C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\directory.
3. (Optional) On the **View** ribbon tab, **3D Rendering** tool in the **Windows** group.
   This view is useful to quickly identify the orientation of the columns with respect to the framing beam members. Here, it is evident that the beams and brace frame into the column flanges (rather than the webs).
4. Either:
   Select the **Run Analysis** tool in the **Analysis** group on the **Analysis and Design** ribbon tab.
or

Press <CTRL+F5>

The STAAD Analysis and Design dialog opens.

During the analysis (and design, if specified), an output file is generated. This file may contain selected input data items, results and error messages. Optional print specifications can be used to include additional information in the output file.

5. Click Done.

6. Select the Connection Design workflow.
   The RAM Connection - Validation dialog opens.

7. Click Close.

EX. To design the roof beam to column connections

1. Hold <Ctrl> and select both upper columns and the roof beam (members 3, 4, and 6).

2. On the Connection Design ribbon tab, select the Smart Connection tool in the Assign Connections group.
3. Select AISC 360-10 (ASD) as the Design Code. Leave all other options unchecked.

4. Select Clip Angle BCF (Clip Angle, Beam-Column(Flange)) from the connection types drop-down list.

5. Double-click DA BCF All bolted in the Available list. It is added to the Selected list.

6. Click OK. 
The RAM Connection - Validation dialog opens to display the design results overview.

7. Click Close. The connection design is added to the RAM Connection Input table.

**EX. To design the floor beam to column connection**

1. Hold `<Ctrl>` and select both the left column sections and the floor beam (members 1, 3, and 5).

2. On the Connection Design ribbon tab, select the Smart Connection tool in the Assign Connections group.

   ![Diagram of a floor beam to column connection]

   The Smart Connections dialog opens.

3. Select AISC 360-10 (ASD) as the Design Code. Leave all other options unchecked.
4. Select Unstiffened Seated BCF (UnStiffened Seated, Beam-Column(Flange)) from the connection drop-down list.

5. Double-click BCF Bolted in the Available list.
   It is added to the Selected list.

6. Click OK.
   The RAM Connection - Validation dialog opens to display the design results overview.

7. Click Close.
   The connection design is added to the RAM Connection Input table.

EX. To design the brace gusset plate connection

So far, you have selected members which form a joint for design. In this procedure, you will use some of the quick selection tools in the program which can be used to select all joints of a similar type.

1. On the Connection Design ribbon tab, select the Select Joints > Select all Column-Beam-Brace Joints tool in the Assign Connections group.
   Members 1, 2, 4, 5, and 7 are selected.

2. Hold <Ctrl> and then click the lower, left column section to remove it from the selection set.
   The lower end of the brace ends at a support so a separate connection design incorporating the base plate is needed.

3. On the Connection Design ribbon tab, select the Gusset Connections tool in the Assign Connections group.
   The Gusset Connections dialog opens.
4. Select AISC 360-10 (ASD) as the Design Code. Leave all other options unchecked.
5. Select Gusset Plate CBB (Gusset Plate, Column-Beam-Brace) from the connection types drop-down list.
6. Double-click the CBB_DW_CBF entry in the Available list. This is a Column-Beam-Brace connection, directly welded, at the column and beam flanges. It is added to the Selected list.
7. Click OK. The RAM Connection - Validation dialog opens to display the design results overview.
8. Click Close. The connection design is added to the RAM Connection Input table.

EX. To design the gusset base plate connection

1. Hold <Ctrl> and select the lower-left column segment and the brace (members 1 and 7).

3. Select AISC 360-10 (ASD) as the Design Code. Leave all other options unchecked.
4. Select Gusset Base Plate from the connection types drop-down list.
5. Click [>>] to include all options in the Selected list. In this case, there is only a single Gusset BP type.
6. Click OK.
   The RAM Connection - Validation dialog opens to display the design results overview.

7. Click Close.
   The connection design is added to the RAM Connection Input table.

EX. To change detailing of a connection

1. In the RAM Connection Input table, double-click the row for connection number 3 (connection BCF - N(3) - M(1,5)).

Tip: You can also use the Joint Cursor tool in the Connection Design ribbon tab to double-click on a joint in the View window.

The Connection Pad window opens. This window contains the details of the current connection design as well as a 3D rendering.

2. In the right details pane, select the Seat angle in the Unstiffened seated section.
   The default design is a relatively small seat angle and larger top angle.

3. Select an L40408 for the Seat angle section and then click OK.
The design, connection detailing, and drawing update immediately.

4. Click the **Results** tool to display the design checks report.

5. In the Report window, select the show Formulas tool.
   The report updates with the calculations for each check.

   **Tip:** You can export either format to a Microsoft Office Word document. Including formulas significantly increases the length of the report.

6. Select the **Exit** tool to close the report window.
7. Select the **Save** tool.
8. Click the **[X]** to close the **Connection Pad** window.
   The design is updated in the **RAM Connection Input** table.

**EX. To generate a report of the connection designs**

1. On the **Connection Design** ribbon tab, select the **Connection Report** tool in the **Reports** group.

   The **RAM Report Export** dialog opens.
2. Select the **Grouped by Identical Connections** options. The two BCF connections for the roof beam are grouped together.

3. Click **Select All**. Leave the other options their defaults: include Data Report, Results Report with Formula, and in a merged reports format.

4. Click **OK**. The report is generated and saved to a Microsoft Office Word document (*EXAMP07_report.doc*) in the same folder as the STAAD input file.

5. Click **OK**.

**Tip:** You can also add a summary table of the connection design results to your STAAD report.

**EX. Connection Tags Example**

This example demonstrates applying and checking connection tags.

**Connection Tag Data**

Use the **Copy to clipboard** button below to copy and paste this connection data into an empty text file and then save it as *Connection_Tag_Example.xml* to a convenient location.

**Tip:** The default location for connection tag data in STAAD.Pro can be found at `%LOCALAPPDATA%\Bentley\Engineering\STAAD.Pro CONNECT Edition\Default\Plugins\ConnectionTagLink`.

```xml
<?xml version="1.0" encoding="utf-8" ?>
<ConnectionTagFile>
  <FileVersion value="1.0" />  
  <Categories>
    <Category CategoryName="MOMENT">
      <CategoryDesc>End Moment Connection</CategoryDesc>
    </Category>
    <Category CategoryName="SHEAR">
      <CategoryDesc>Single Shear Connection</CategoryDesc>
    </Category>
  </Categories>
  <Equations>
    <Equation EquationID="Eq1" Equation="(abs([Mz])+([alpha]*abs([Fx])))/[Mz.cap]" Condition="LT" Limit="1.0" />
    <Equation EquationID="Eq2" Equation="abs([Fy])/[Fy.cap]" Condition="LT" Limit="1.0" />
    <Equation EquationID="Eq3" Equation="(abs([Fy])+([alpha]*abs([Fx])))/[Fy.cap]" Condition="LT" Limit="1.0" />
  </Equations>
  <Tags>
    <Tag TagName="EM" CategoryName="MOMENT">
      <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
      <Capacities UnitSystem="IMPERIAL">
        <Beam Name="W12x26">
          <BeamOrCol Name="W14x90" Mz.cap="700" Fy.cap="65" Fx.cap="" alpha="0.10" />
        </Beam>
      </Capacities>
    </Tag>
  </Tags>
  <Checks>
    <ApplicationExamples>
      <EX. Steel Design Examples>
    </ApplicationExamples>
  </Checks>
</ConnectionTagFile>
```
<Check Type="MOMENT" Desc="Moment Check" EquationID="Eq1" />  
<Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
</Checks>
</Tag>
<Tag TagName="EMH" CategoryName="MOMENT">
  <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
  <Capacities UnitSystem="IMPERIAL">
    <Beam Name="W12x26">
      <BeamOrCol Name="W14x90" Mz.cap="700" Fy.cap="65" Fx.cap="" alpha="0.10" />
    </Beam>
  </Capacities>
</Tag>
<Tag TagName="SS" CategoryName="SHEAR">
  <EndRelease FX="0" FY="0" FZ="0" MX="1" MY="1" MZ="1" />
  <Capacities UnitSystem="IMPERIAL">
    <Beam Name="W12x26">
      <BeamOrCol Name="Column" Mz.cap="" Fy.cap="65" Fx.cap="" alpha="1" />
      <BeamOrCol Name="W14x90" Mz.cap="" Fy.cap="65" Fx.cap="" alpha="0.75" />
    </Beam>
  </Capacities>
</Tag>
<Tag TagName="DS" CategoryName="SHEAR">
  <EndRelease FX="0" FY="0" FZ="0" MX="1" MY="1" MZ="1" />
  <Capacities UnitSystem="IMPERIAL">
    <Beam Name="W12x26">
      <BeamOrCol Name="Column" Mz.cap="" Fy.cap="65" Fx.cap="" alpha="1" />
      <BeamOrCol Name="W14x90" Mz.cap="" Fy.cap="65" Fx.cap="" alpha="0.75" />
    </Beam>
  </Capacities>
</Tag>
</Tags>
</ConnectionTagFile>

**Note:** This is an extremely short connection data example for the purposes of demonstration. A typical connection data file would contain a very large number of connection capacities for checking.

**EX. To open the example and load example connection data**

Copy and save the example connection tag data presented in the previous topic before proceeding.

1. Open the file example file **US.7. Modeling Offset Connections in a Frame.std** in the **Analytical Modeling** workflow.
This file is typically found in the
C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\directory.

2. On the Utilities ribbon tab, select the Connection Tags > Open Tab Specifications tool in the Tools group.

The Select Connection Tag File dialog opens.

3. Navigate to the Connection_Tags_Example.xml file and click Open.

This file can be found where you saved it in the previous topic.

EX. To assign the connection tag types to beam ends

1. Select both beams 5 and 6.
   
   Tip: Press <Shift+B> to display the beam numbers.

2. On the Beam Tools ribbon tab, select the Assign tool in the Connection Tags group.

   The Assign Connection Tags and New Connection Tag dialogs open.

3. In the New Connection Tag dialog, select the options to generate the tags:
   
   a. Select EM from the Select Tags drop-down list.
   b. Select both Start and End options for the Location.
   c. Click Assign.

   The dialog closes and the two tags are added to the Assign Connection Tags dialog, assigned to the members.

   Note: There are two tags generated, one for the beam start and another for the beam end.

4. Click Close on the Assign Connection Tags dialog.

EX. To check the connection tags

1. On the Utilities ribbon tab, select the Connection Tags > Check Tags tool in the Tools group.

   The Assign Connection Tags and Check Connection Tags dialogs open.

2. In the Check Connection Tags dialog:
   
   a. Check the option for the 1 - WIND LOAD load case.
   b. Click Check.

   The Connection Tag Check Results dialog opens.
3. (Optional) Click the Print tool to open the print Preview window, where you can save a copy of the connection tag results table.

EX. Interactive Concrete Design Examples

This is an example of using the interactive Concrete Design workflow to design an example concrete structure.

EX. BS8110 British Code Design Example

This example uses the STAAD.Pro UK example 9 model, which can be found in the STAAD.Pro installation folder at C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK.9. Modeling Slabs and Shear Walls Using Finite Elements.STD.

The following outlines the steps to create the model and element designs.

EX. BS8110 Model Setup

The following procedures are used to open an example model, select loads to add to a design envelope, setup up design briefs including code parameters, and add members to a design group.

EX. To open the example model in RC Designer

Before carrying out any specific design, first perform the following steps which are general to all the designs:

   
   This file is typically installed with the program in C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\.
2. On the Analysis and Design ribbon tab, select the Run Analysis tool in the Analysis group.

This produces a set of analysis results required for design.

3. Select the Concrete Design workflow.

The model is opened in Concrete Design.

4. (Optional) Enter job information in the Job Information dialog for inclusion in reports.

EX. To create the envelope used in the beam design

1. Select the Design Layer | Envelopes page (on page 1202) in the page control found on the left side of the program window.
2. Click New Env. at the bottom of the Envelopes Table (on page 1204) to create a defined envelope of load cases and combinations.
   The New Envelopes dialog (on page 1204) opens.
3. Enter the name Envelope 1 in the E# field.
4. Click OK.
   The Define Envelopes dialog (on page 1205) opens, which allows the setting of which load cases and combinations are to be considered for Envelope 1.
5. De-select the option to Show Combinations Only.
   Primary load cases L1 and L2 are now displayed in the Load Cases list.
6. Click the >> button
   All the load cases and combinations L1, L2, and C3 are now included in the envelope E1.
7. Click OK to accept this definition.

EX. To create the design members

1. Select the Design Layer | Members Page (on page 1202) in the page control.
2. Either:

   Use the Elements Cursor tool (selected by default) to drag a box around the entire model.
   or
   Select Select > Select All.
3. Select Members > AutoForm Members to automatically create design members.

   Fifteen members are created.

   Note: The horizontal beams and corner columns have been created correctly, but the center columns have been formed into small segments (M8, M9, M10, M11 and M12, M13, M14, M15) which are not ideal physical members for design.
4. Select the Members Cursor tool.
5. Select these unconnected members (see note above) and press the Delete key.
6. Select the Elements Cursor tool
7. Select the four elements that make up the column on one side
8. Select Members > Form Member (on page 1150).
   A new, single member, MB, is created.
9. Repeat steps 6 through 8 for the four elements that form the column on the other side to form member M9.

EX. To create a floor slab

1. Select the Slabs Page in the page control.
2. Select the Side View tool to get an elevation view of the model.
3. Use the Select Plates tool (selected by default) to drag a window to select all the plates that form the top floor.
   Tip: Be careful not to select the plates that form the top of the shear wall.
4. Select Slabs > Form Slab.
   Tip: Note the details of the slab in the Slabs table (on page 1153) and confirm that it has been created with 32 plates and has no holes. If not, delete the slab definition and repeat steps 3 and 4 until the correct slab is created.

EX. To create a BS 8110 beam design brief

1. Select the Groups/Briefs Page in the page control.
2. Click New Brief at the bottom of the Design Briefs table (on page 1155).
   The New Design Brief dialog (on page 1156) opens.
3. Enter BS8110 Beam Brief in the B1 field.
4. Select BS8110 in the Design Code list.
5. Select Beam in the Design Type list.
6. Click OK.
   The BS8110 Beam Brief dialog (on page 1255) opens.
7. Select C40/50 for the Concrete Grade.
8. Select the Main reinforcement tab
10. Click OK.

EX. To create a BS 8110 slab design brief

1. Click New Brief at the bottom of the Design Briefs table (on page 1155).
   The New Design Brief dialog (on page 1156) opens.
2. Enter BS8110 Slab Brief in the B3 field.
4. Select Slab in the Design Type list.
5. Click OK.
   The **BS8110 Slab Brief** dialog (on page 1260) opens.
6. Select **C40/50** for the **Concrete** grade.
7. Click OK.

**EX. To create a beam design group**

1. Use the **Members Cursor tool** to select the five members that form the perimeter of the slab and the member that lies along top of the shear wall in the center of the model (M1, M2, M3, M4, and M5).
   
   **Tip:** Hold <CTRL> to add members to the selection set.
2. Click **New Design Group** on the **Design Groups table** (on page 1160).
   The **New Design Group dialog** (on page 1149) opens.
3. Enter **BS8110 Beam Group** in the **G1** field.
4. If **BS 8110 Beam** is not the selected **Design Brief**, then select it.
5. Click OK.
6. Double click the G1 group in the Design Group table.
   The **Group Member list** (on page 1160) opens for the selected Design Group.
7. Confirm that the design group contains the correct list of design members.

**EX. To create a column design group**

1. Use the **Members Cursor tool** to select the six members that form the columns of the slab, including the two internal columns that edge the internal shear wall (M6, M7, M8, M9, M16, and M17).
   
   **Tip:** Hold <CTRL> to add members to the selection set.
2. Click the **New Design Group** button on the **Design Groups table** (on page 1160).
   The **New Design Group dialog** (on page 1149) opens.
3. Enter **BS8110 Column Group** in the **G2** field.
4. Select **BS 8110 Column** in the **Design Brief** list.
5. Click OK.
   The **Group Member list** (on page 1160) opens for the selected Design Group.
7. Confirm that the design group contains the correct list of design members.

**EX. To create a slab design object**

1. Select the Slabs tab at the top of the **Design Groups table** (on page 1160).
2. Click the New Design Slab button.
   The **New Design Slab dialog** (on page 1162) opens.
3. Enter **BS8110 Slab** in the **SD1** field.
4. If **BS 8110 Slab Brief** is not the selected **Design Brief**, then select it.
5. Select 1 in the **For Slab** list.
6. Click OK.

The model is now ready for performing design.
Tip: Now is a good time to save your model if you haven't already done so.

EX. BS8110 Beam Design

The model is ready for the beams in Design Group 1, G1: BS8110 Beam to be designed.

EX. To design the beam members in the concrete member mode

1. Select the Concrete Member mode in the page control. The Concrete Member | Summary page opens.
2. Ensure that the current design group is **G1: BS8110 Beam** and, if not, select it from the drop list on the Page/Mode Automatic toolbar (on page 1198). Note how Members M4 and M5 are identified as two spans in the Beam Group Summary and Beam Spans tables.
   Tip: This can changed by changing supports definition of the nodes in the on the Beam Spans table (on page 1177) Supports tab.
3. Select the Concrete Member | Design Page in the page control. The Design Options dialog (on page 1168) opens.
   Note: Members in the current design group that can be designed are automatically included, but some can be removed if no design is needed.
4. Click Design. All five members are displayed in green and the Summary table (on page 1187) shows the status for parts of all five beams as OK.

EX. To view the reinforcement layout for member 4

1. Select the Concrete Design | Main Layout Page in the page control.
2. Select **M4** (Member 4) from the design group members drop list found on the Page/Mode Automatic toolbar (on page 1198). The reinforcing details and moment envelope diagrams update for this member.
3. In order to see the full extent of the beam, drag the horizontal separator bar below the elevation view of the beam up until it looks like the following:
EX. To view the calculations

2. On the Items tab, select Design Detail in the Available list and move it to the Selected list by clicking the > button.
3. Select the Members tab.
4. Click the << button to un-select all the members.
5. Select Member 4 and click the > button to move it to the selected side.
6. Click **OK** to accept this Report Setup.
7. Select **File > Print Preview** to view the resulting report.
   The **Print Preview window** (on page 1115) opens.

The detail calculations will be displayed approximately on page 5 on a report of 14 pages:
Application Examples

EX. Interactive Concrete Design Examples

Member M4 Span 1
Detailed BS8110 Design Requirements

Section Property: 200 x 299
Span Length = 6.000 m Rectangular section
Width = 300 mm Depth = 300 mm
Covers: Top = 30 mm Bottom = 30 mm Side = 30 mm

Member M4 Span 1
Detailed BS8110 Main Reinforcement

Hogging: at 0.000 m from the start of the member

Moment applied to section
Effective depth of tension reinforcement
Depth to compression reinforcement
Redistribution < 10%, hence

\[ K = \frac{M}{bd^2 f_y} \]

\[ K \leq K_c \text{ hence compression steel not required.} \]

\[ z = d \left( 0.5 + \left( 0.25 \cdot \frac{K_c}{K} - \frac{K}{0.9} \right) \right) \leq 0.95d \]

\[ A'_{w} = \frac{0.87 f_z}{M} \]

Tension Bars provided = 2H16
Actual area of tension reinforcement
Minimum area of tension reinforcement
Maximum area of tension reinforcement
Actual% of tension reinforcement

4943 User Manual
**EX. BS8110 Column Design**

The model is ready for the columns in Design Group 2, G2: BS8110 Column to be designed.

**EX. To design the column members in the concrete member mode**

1. If you are not still in the Concrete Member mode, select it in the page control.
2. Select **G2: BS8110 Column** as the current design group from the drop list on the Page/Mode Automatic toolbar (on page 1198). The Concrete Member | Summary page (on page 1207) opens.
3. Select the Concrete Member | Design page (on page 1207) in the page control.
   The Design Options dialog (on page 1168) opens.

   **Note:** Members in the current design group that can be designed are automatically included, but some can be removed if no design is needed.

4. Click the **Design** button to design all six members.

   **Note:** This loading is not substantial and thus results in a Design Messages dialog which suggests that though these are axially loaded members, they are subject to greater bending moments and should be considered as beams. However, they will suffice for this example.

   All six members should be displayed in green and the Summary table (on page 1187) shows the status for parts of all six columns as OK.

**EX. To view and edit the reinforcement layout for member 6**

1. Click on the Concrete Member | Main Layout page (on page 1207).
2. If it is not already selected, select Member 6 from the drop-down of members in the current design group.
   The initial design produced an arrangement with the nine T16 bars in the locations shown in the diagram above. It is possible to replace each of the bars in the center of each face of the column with two smaller T8 bars.
   The Main Reinforcement window displays a force diagram, reinforcement elevation, and reinforcement section of the current design.
3. Right-click anywhere on the cross section (right pane).
4. Select Edit Bar Cage from the pop-up menu.
   The M# Bar Positions dialog (on page 1181) opens.
5. Set the distance for the Top and Side bars to 3.0cm
6. Reduce the size to TBs by clicking in the Size cell and selecting 8 from the drop-down list.
7. Click the Apply button.
   The bar arrangement diagram is updated.
8. Select **Design > Design Now** to check this arrangement.

    **Note:** While the same warnings appear about the small amount of axial load, the design shows that the arrangement is adequate.

**EX. To view the calculations**

1. Select **File > BS8110 Report Setup**.
2. On the Items tab, select Design Detail and move it to the Selected list by clicking the > button.
3. Select the Members tab.
4. Click the << button to un-select all the members
5. Select Member 6 and click the > button to move it to the selected side.
6. Click the OK button to accept this Report Setup.
7. Select **File > Print Preview** to view the resulting report.

   The detail calculations will be displayed approximately on page 2 on a report of 3 pages.

**EX. BS8110 Slab Design**

The model is ready for the slab SD1: BS8110 Slab to be designed.

**EX. To view displacement patterns in the concrete slab mode**

1. Select the Concrete Slab mode in the page control.

   The **Concrete Slab | Summary page** (on page 1212) opens.

   **Note:** This model only contains a single slab. If it contained more than one, then the required slab should be selected from the toolbar drop list.

2. Select the **Concrete Slab | Displacements page** (on page 1213).
3. Select **Results Line > Unconstrained**.
4. Select **Result Line > Create**.
   The mouse pointer changes to a Result Line drawing cursor in the slab diagram.

5. Click near the top center of the slab.
   A dashed line connects this point to your mouse pointer, indicating where the result line will placed.

6. Click a second point near the bottom center to complete the result line.
   The result line is drawn as a solid, black line with an arrow indicating the positive direction along the line.
   The Result Line Graph and table display the displacement along this line.

**EX. To define slab design regions**

1. Click on the **Concrete Slab | Regions** page.
   The slab defaults to a single boundary.

2. Select **Regions > Draw Boundaries**.
   The mouse pointer change to a Draw Regions cursor in the slab diagram.

3. Click the center node (top of the middle column) on the left-hand side of the diagram.
A dashed line connects this point to your mouse pointer, indicating where the region boundary will be placed.

4. Click the center node (top of the second middle column) on the right-hand side of the diagram. Two regions are created, one on either side of the newly created boundary line. The Regions table (on page 1166) indicates the details for each region, including the orientation.

5. Rotate the axes of region 1 by changing the orientation to 90 degrees in the Regions table.

**Note:** The orientation angle is taken counter-clockwise (i.e., anti-clockwise).

6. Select the Concrete Slab | Design page.
   The Design Options dialog box (on page 1214) opens.

7. Click the Design button to perform the design.
   The slab diagram displays a contour of the Spacing in the X direction along the top slab surface.

8. Select Bottom:Y direction and the Reqd Steel result in the toolbar.
   This will show a contour line pattern. The contour lines can also be displayed as filled to aid in viewing results.

9. Either:
   Select View > Diagram.
or

Right click on anywhere on the diagram and select **Diagram**... from the pop-up menu.
The **Diagrams dialog box** (on page 1144) opens.

**10.** On the Design Contour tab, select the option for **Filled** style and then click **OK**.

**EX. To print a design report**

1. **Select File > BS8110 Report Setup**...
The **BS8110 Report Setup** dialog box (on page 1110) opens.
2. **Confirm that the report contains the items:**
   - Job Information
   - Slab Brief Detail
   - Slab Information
   - Region Information
   - Design Information
3. **Click OK**
4. **Select File > Print Preview.**
The report is fitted on approximately two pages, with the second page containing the design results.

**EX. Advanced Concrete Design Tutorial**

**Getting Started**

STAAD Advanced Concrete Design is performed using the RCDC application, which is installed separately from STAAD.Pro. This program can be launched from the STAAD.Pro Building Designer mode or as a standalone application.

RCDC is a highly automated design tool for concrete building structures. This tutorial guides you through the process of designing the elements in a concrete building. It will also indicate where to change the design settings to customize design and detailing to meet the requirements of your project or organization.

**Tutorial File**

This tutorial uses example files that are installed with STAAD.Pro V8i. However, you will be making changes to this example file so you may wish to make a backup copy to preserve the example as installed with STAAD.Pro. A copy of the original input file is also included as **Appendix A** (on page 4979) of this tutorial.

**Starting in STAAD.Pro**

This example includes a fully modeled concrete structure, including loads. The modeling and application of loads per a building code specification are outside the scope of this tutorial. Refer to the STAAD.Pro help contents for additional information.
Using RCDC as a Standalone Application

You can also use RCDC as a standalone application for concrete design. You will still need to have a analytical model created in STAAD.Pro for the basis of the physical model as well as for analytical results.

Design and detailing of foundation elements such as footings and pilecaps must be performed by starting RCDC as a standalone application and creating a project associated with an analytical model.

To open the model and generate analysis

Tip: It is recommended that you first make a backup copy of all files with the name Typical Ground +3 Story Building in the C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Building Planner\folder. You can do this using the Archive feature (on page 2272) in STAAD.Pro.

1. Open the STAAD input file
   C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Building Planner\Typical Ground +3 Story Building.std

   Tip: If you have a ProjectWise Project you want to associate with this input file, you can do so. This is not necessary, though, for this tutorial.

2. (Optional) On the File ribbon tab Info tab, add job information:
   - type Tutorial in the Job field
   - type your organization’s name in the Client field
   - type your initials in the Engineer field

3. Either:
   - on the Analysis and Design ribbon tab, select the Run Analyze tool in the Analysis group
   - press <Ctrl+F5>

   The analysis is performed.

4. Select the Stay in Modeling Mode option and click Done.
5. Select the Building Planner workflow. The STAAD Building Planner window opens.

The STAAD Building Planner will be used to layout the structure.

To create a building model from floor plan

In order to demonstrate column design in RCDC, you will use the Building Planner model rapidly model a building from a single floor plan.

1. Select the Frame | Geometry page.
2. Click Edit in the Level Details dialog. The Level Information dialog opens
3. Type 6 (m) in the Height cell for row No 0-1 and press <Enter>.
4. Click Add Level and then click Yes in the confirmation dialog. A new row is added representing a new level.
5. Select Plan1 from the Plan drop-down list for the second level. Leave the default heights and other values for this level.
6. Click OK.
7. In the Level Details dialog, click Generate Model and then click Yes to confirm you want to update the STAAD input model. The Space Frame File Generation dialog opens.
8. Click **Generate**.
   For this tutorial, you will use the default settings, load settings, and building type. Refer to the STAAD.Pro User Interface help for details on these settings.
   The new input file is generated and opened.

9. Either:
   - select **Analyze** > **Run Analysis**
   - or
   - press <Ctrl+F5>
   The analysis is performed.

10. Select the **Stay in Modeling Mode** option and click **Done**.

To add a shear wall

To add a shear wall along one side of the elevators, do the following.

For the purpose of this tutorial, you will only add a single shear wall. In a typical building, other shear walls would be present.

1. In the Building Planner mode, select the **Frame | Shear Wall** page.
2. In the **Shear Wall Details** dialog, click **Edit**.
3. Click on column C31 and then on C32.
The Define Shear Wall dialog opens.

4. Click OK.
   For this tutorial, use the default thickness. The shear wall is added on both levels using this plan by default.

You have now finished all the necessary modifications to this example for this tutorial. The next sections cover the design and detailing of structural elements using RCDC.

**Slab Design**

This section of the tutorial demonstrates how to design and detail a slab, including openings and stairs.

Slab continuity is based on edge conditions where are automatically determined by the program. You can manually change the end conditions of a slab. You can also change a slab to a cutout, raised slab, or depressed slab which also affect the edge conditions.

Slab properties are imported from the analysis model where specified. The program will evaluate missing properties (e.g., slab thickness) during design when not specified.

To start the slab design

**Tip:** When you import STAAD.Pro model data into RCDC, grids are automatically detected and assigned. You can manually edit both the exact locations of grids as well as grid labels in RCDC for drawing generation.

1. In the top-right dialog, select **Level:1** from the drop-down list and then click **Design Slab**.
RCDC opens and the **New Project** dialog opens.

2. Enter the project data:

![New Project dialog](image)

- **a.** Type **Tutorial** in the **Project** field.
- **b.** (Optional) Type **Client** and **Engineer** name.
  
  This information is used as the header for all reports. It will be automatically imported from the STAAD.Pro input file Job Information block if available.
- **c.** Select **EN 02 - 2004** from the **Design Code** drop-down list.
  
  This tutorial uses the Eurocode. However, you can select any available code. Just note that some design settings and features may vary based on the code selection made here.

The slab edges are automatically detected and continuity is established.
3. Either:

select the **Design Parameters** tool

or

select **Settings > Design Parameters**

The **Design Settings** dialog opens.
4. In the Available Rebar list, clear the check mark by 25 (mm). This tutorial will limit the rebar size.

   **Note:** All other design settings are left as their defaults, but you can make many design and detailing changes here.

5. Click **OK**.
6. Either:

   select the **Crack Width Setting** tool

   or

   select **Settings > Crack Width Settings**
   
   The **Crack Width Settings** dialog opens.
7. Select the **Perform Check** option in the **Crack Width** group.

   **Note:** Leave the Permissible Crack Width as the default value and leave the Perform Check option in the Initial Thermal Cracking group unchecked for this tutorial.

8. Click **OK**.

   **To specify elevator openings**

   Since the analysis model does not contain openings in the slab, you can add those in RCDC.

   **Note:** RCDC can design and detail concrete stairs with the slab. Do *not* create an opening for the stairs if you plan to use this feature. Refer to the RCDC help topic **Slab > Staircase Section** for details on this feature.

1. In the Design Data table, select row 19 (for Slab 18).
   The slab is highlighted in the Slab Layout view window.
2. Do either of the following:
   - select **True** in the **Is Cut Out** cell for this row in the table
   - or
   - right-click on the slab in the view window and select **True** from the **Is Cut Out** drop-down in the pop-up menu
The slab is now marked as a cutout.

3. Select Modify > Re-evaluate Slab continuity.

**Note:** This will change the continuity of surrounding slabs. If you do not perform this step, you will be prompted to automatically re-evaluate the continuity before performing an autodesign.

To perform the design

Slabs in RCDC are designed for the **Imposed Load** and **Live Load** values input for each slab on the **Design Data** table. These values are imported (along with thickness, etc.) from the slab parameters specified in STAAD.Pro Building Planner mode.

1. Either:
   
   select the **Autodesign** tool.

   or

   select Design > Autodesign.

   A message dialog opens confirming you want to proceed since the continuity must be reevaluated if you have not already done so.

2. If you did not previously reevaluate the continuity, click **Yes**.

   The Design Output page opens.

   A warning message opens stating that two of the slabs have failed the design checks. The same slab is marked with a red row number in the table.

3. Select the row for the first failed slab (row 13, S24) in the table and then either:

   select the **Design** tool.

   or

   select Modify > Design

   or

   right-click and select Design from the pop-up menu
The **Design Slab** dialog for this slab opens at the bottom of the program window.

4. Type 325 (mm) in the **Thickness** field in the **Design Parameters** group.

5. Select the **Design** tool in the **Design Slab** dialog.
   The slab now passes the design checks.

6. Select the **Accept** tool to update the design with the changes.

7. Repeat steps 3 through 6 for S17 (row 24) with a thickness of 200 (mm).

To generate construction and detail drawings

1. Generate a GA drawing for the slab:
   a. Select **Reports > General Arrangement Plan**.
      The **GA Drawing** view window opens.
   b. Click the **Save** tool in the view window.
   c. Type **Sample_Slab_GA.dxf** and click **Save** to save the GA drawing as a DXF file.

2. Generate a schedule:
   a. Either:
      select the **Text Schedule** tool
   or
      Select **Reports > Text Schedule**.
      A dialog opens to ask if you want to group similar slabs.
   b. Click **Yes**.
      The **Text Schedule** view window opens.
   c. Click the **Save** tool in the view window.
   d. Type **Sample_Slab_Sched.dxf** and click **Save** to save the schedule as a DXF file.

3. Generate the reinforcement layout drawings:
   a. Either:
      select the **In-Plan Detailing** tool
   or
      Select **Reports > In-Plan Detailing**.
      The Top Reinforcement and Bottom Reinforcement drawings open.
   b. Click the **Save** tool in the view window.
   c. Type **Sample_Slab_TopReinfPlan.dxf** and click **Save** to save the schedule as a DXF file.
   d. Repeat steps 4b and 4c to save the bottom reinforcement plan as **Sample_Slab_BotReinfPlan.dxf**.

4. Generate a bill of quantities:
   a. Either:
      select the **Bill of Quantities** tool
   or
      Select **Reports > Bill of Quantities**.
      The **Bill Of Quantities** dialog opens.
b. Select the BOQ Summary option.
c. Select € - Euro from the Currency drop-down list.
d. Type 90 (€/m$^3$) for Grade C20/25 concrete and 4 (€/kg) for Grade Fy420 steel. Type 8 (€/m$^2$) for the Shuttering value.
e. Click OK.
   The BOQ Summary opens.
f. Either:
   - click the Print tool to print out a copy of this report
   - or
   - click the Save tool to save an HTML copy of this report

To generate design calculation reports

1. Either:
   - select the Design Summary tool
   - or
   - select Reports > Design Summary
   The Slab Design Summary opens.
2. Either:
   - select the Design Calculations tool
   - or
   - select Reports > Design Calculations
The Select Slabs dialog opens.
3. Check the boxes for S1 and S12.
   For this tutorial you will only output the detailed design for two slabs. You can check the All Slabs option to include the full design report.
4. Click OK.
   The Slab Design opens.
5. Either:
   - Click the Print tool to print out a copy of this report
   - Click the Save tool to save an HTML copy of this report

To save the project and exit
1. Either:
   - Select the Save tool (on the main program toolbar)
   - Select File > Save
   The Save As dialog opens.
2. Type Sample1_Slab_tutorial.rcdx in the filename and click Save.
   RCDC closes and you are returned to STAAD.Pro.

RCDC Settings Files
When you save your project in RCDC, in addition to the project file (.rcdx) a project settings file is also created (.rcps). You can select File > Import Project Settings to open the project settings for a previous project once you have established project or office standards you want to re-use.

Beam Design
This section of the tutorial demonstrates how to design and detail continuous concrete beams.
RCDC will automatically look for continuous beam members in the STAAD model and group these as a single physical beam. This way if you have a beam which continuous through columns or one which was sub-divided into STAAD.Pro to facilitate support of a transverse member, it will be designed as a single beam entity in RCDC.
In this example, you will use most of the default values for all designs.

To start the beam design
In STAAD.Pro, ensure you are on the Frame | Design page in the Building Planner mode.

   **Tip:** When you import STAAD.Pro model data into RCDC, grids are automatically detected and assigned. You can manually edit both the exact locations of grids as well as grid labels in RCDC for drawing generation.

1. In the top-right dialog, check the Level :1 option and then click Design Beams.
RCDC opens and the New Project dialog opens.

2. Enter the project data and select the design code:

   a. Type Tutorial in the Project field.
   b. (Optional) Type Client and Engineer name.
      This information is used as the header for all reports. It will be automatically imported from the STAAD.Pro input file Job Information block if available.
   c. Select EN 02 - 2004 from the Design Code drop-down list.
      This tutorial uses the Eurocode. However, you can select any available code. Just note that some design settings and features may vary based on the code selection made here.

The beam continuity is automatically detected as the analytical model data is read.

3. Either:

   select the General and Reinforcement Settings tool
   or

   select Settings > General and Reinforcement Settings

The General and Reinforcement Settings dialog opens.
4. In the **Rebar** list, clear the check mark by both 32 (mm) and 40 (mm). The initial design will be evaluated using smaller size bars where possible.

   **Note:** All other design settings are left as their defaults, but you can make many design and detailing changes here.

5. Click **OK**.

   **Note:** There are numerous other detailing and drawing settings in RCDC that allow you to customize the concrete detailing for your client or organization’s needs. For details on these options, refer to the RCDC help by pressing <F1> or selecting **Help > Contents**.

To specify load cases and combinations

1. Select **Settings > Basic Load Cases**
   The **Basic Load Cases** dialog opens.

2. (Optional) If the **Load Type** selection is not already specified for each load case, select it from the drop-down list:
3. (Optional) If both loads are not already added to the Primary Load Cases list, click the \([+]\) button when a load case is selected in the Analysis Load Cases list to add it.

   **Note:** You can import previously saved load cases and combinations to use by clicking the **Import Load Cases & Combinations** button and selecting an .ldsx file. Refer to the following steps on how to create a load settings file.

4. Click **OK**.
   The **Load Combinations** dialog opens.

5. Click **Add From Analysis**.
   The load combination from the analysis file is added for strength design checks.

6. Add the load combination for use in evaluating crack width:
   a. Select the **Crack Width** tab.
   b. Click **Add From Template > For Regular Structure**.
      A service level load combination is added from the RCDC load combinations template.

   **Note:** Typically, a different (service level) load combination is used for evaluating crack widths. If you already have appropriate service load combinations in your analytical model, you can add those instead and select them for use here.

   c. Check the options for **Crack Width** and **Stress Limit** in the load load combination in the table.

7. Export the load cases and combinations for later use:
   a. Click **Export Load Cases & Combinations**.
      The **Save As** dialog opens.
   b. Type **Sample1_Loads** and click **Save**.

8. Click **OK**.

**To split beam group at staircase**

1. Select the **Design Input | Beam Continuum** tab.
2. Select the third beam group in the chart.

   **Note:** The beams on grid line 6 are highlighted in the Layout view.

3. Right-click on the blue box labeled C17 (column 17) and select **Split Group** from the pop-up menu.
   The beam group is split into two physical members at this location.

4. Select the **Design Input | Design Grouping** tab.
5. Select G19 in the table or click on the newly created beam group in the Layout view.
6. Type **-900** in the **Top** field for both beam segments in G19 (B14 and B15).
   This represents a top of beam elevation of 0.9m below the floor elevation.

**To perform the beam design**

1. Either:
   select the **Autodesign tool**

   or
select Design > Autodesign.

An exceptions dialog opens indicating that two beams have failed the design checks.

2. Click OK.

3. Either:

   select the Failure Diagnostics tool

or

select Reports > Failure Diagnostics

The Failure Diagnostics tab opens. The table indicates that many beams are failing due detailing checks.

4. Evaluate some options for beam group 2:

   a. Select any of the beam segments in G2 in the Design Output table.

      The beam segments in this group are marked with a red tag to indicate the group failed one or more checks.

   b. Either:

      select the Design tool

or

select Modify > Design

The Redsign Grp dialog opens at the bottom of the program window.

   c. Type 3 in the Bar Layers at Top field.

   d. Select the Redesign tool in the Redesign Grp dialog.
The beam now passes design and crack width checks.

**e.** Select the **Accept** tool.

The Design Output table now shows green for the G2 beam indicating it passes.

5. Repeat step 4 to redesign G7 with 3 layers of top bars as well. Optionally, you can use the **UnLock** tool to unlock the current design and then change the initial settings to use larger reinforcement sizes.

6. Review the detailing on the demand and capacity curve for a beam:

   a. Select beam group G1 and either:

      - select the **Detail** tool

      or

      - select **Modify > Detail**

      or

      right-click and select **Detail** from the pop-up menu

   The Redtail dialog opens at the bottom of the program window.

   b. Click-and-drag your mouse cursor over the beam subdivisions starting at 0.75L of the first beam span in G1.

   c. Right-click and select **Detail** from the pop-up menu

   The Top Steel pop-up dialog opens.

   d. Delete the N1 value for layer 4 and then select the empty value for D1 for the same layer.

   This will remove the fourth layer of bars from the top steel in this segment, creating a new reinforcement detail for the selected segment.

   e. Click **OK**.

   The section of the capacity curve for the top steel updates to reflect the detailing change.
f. Click Cancel.

**Tip:** You can use select the beam segments or entire reinforcement zones by their labels just below the diagram. You can copy and paste reinforcement details between zones or selections.

7. (Optional) Check for any resized beams during design by selecting **Reports > Beam Size Changed**.
A report including any resized beams during design opens. For this example, none of the beams were resized.

To generate beam schedules and drawings

1. Generate a text schedule of beam reinforcement:
   a. Either:
      select the **Text Schedule** tool
   or
      Select **Reports > Text Schedule**.
      The **Schedule Type** dialog opens.
   b. Select the **Type 1** option.
   c. Select the **Group Beams** option.
   d. Click **OK**.
      The **Text Schedule** view window opens.
   e. Click the **Save** tool in the view window.
   f. Type **Tutorial_Beam_Sched.dxf** and click **Save** to save the schedule as a DXF file.

2. Generate an elevation for any beam group:
   a. In the **Design Output** table, right-click on any beam group and select **Quick Elevation** from the pop-up window.
      The elevation drawing for this beam group opens in the lower section of the program window.
   b. (Optional) Select the **Save** tool in the **Elevation** view to save this view as a DXF file.
   c. Select the **Cancel** tool to close the elevation view

3. Generate the elevation and sections for a beam group:
   a. Either:
      select the **Elevation and Section** tool
   or
      Select **Reports > Elevation and Section**.
      The **Select Beams** dialog opens.
   b. Select group **1.B1-B2-B3-B4** in the beams list.
   c. Select the following Draw options:
Tip: You can modify the drawing and detailing options by clicking the **Detailing & Drawing Settings** button (also by selecting **Settings > Detailing & Drawing Settings** before using this tool). You can also customize rebar mark styles from this dialog.

d. Click **OK**.
The Elevation for the selected beam group opens.
e. Click the **Save** tool in the view window.
f. Type **Tutorial_Beam_Sched.dxf** and click **Save** to save the schedule as a DXF file.

4. Generate a bill of quantities:
a. Either:
   - select the **Bill of Quantities** tool
   - or
   
   Select **Reports > Bill of Quantities**.

   The **Bill Of Quantities** dialog opens.

   ![Bill Of Quantities dialog](image)

   b. Select the **BOQ Summary** and **Reinforcement Type Wise** options.
   c. Select **€ - Euro** from the **Currency** drop-down list.
   d. Type 90 (€/m³) for Grade C20/25 concrete and 4 (€/kg) for Grade Fy420 steel. Type 8 (€/m²) for the Shuttering value.
   e. Click **OK**.

   The **BOQ Summary** and **BOQ Detailed - Reinf Type Wise** tabs open.
f. Either:
click the **Print** tool to print out a copy of this report

or

click the **Save** tool to save an HTML copy of this report

5. Generate bar bending schedule:

   a. Select **BBS > Generate BBS**.
      The **Generate BBS** dialog opens.

   b. Select **1.B1-B2-B3-B4** and **2. B5-B6-B7-B8-B9** in the list of beam groups.
   c. Select the **Continuous** option in the **Rebar Numbering** group.
   d. Select both the **Drawing** and **Spreadsheet** options in the **Schedule Format** group.
   e. Click inside the file path field, type **Tutorial_Beam_BBS** in the **Save As** dialog that opens, and then click **Save**.
   f. Click **Generate**.
      The BBS opens containing a text schedule of the bar bending schedule. If you have Microsoft® Office Excel® installed (or another spreadsheet program capable of opening .xslx format spreadsheet files), the spreadsheet of the bar bending schedule opens.
   g. Close your spreadsheet application.
   h. Click the **Save** tool in the view window.
   i. Type **Tutorial_Beam_BBS.dxf** and click **Save** to save the schedule as a DXF file.

To generate beam design calculations

1. Either:
   - select the **Design Summary** tool
   or
   - select **Reports > Design Summary**
      The **Design Summary** opens.
2. Either:
select the Design Calculations tool

or

select Reports > Design Calculations
The Select Beams opens.

For this tutorial you will only output the detailed design for two beam groups. You can check the All Beams option to include the full design report.

4. Click OK.

5. Either:
   click the Print tool to print out a copy of this report
   or
   click the Save tool to save an HTML copy of this report

To save the project and exit

1. Either:
   select the Save tool (on the main program toolbar)
   or
   select File > Save
   The Save As dialog opens.

2. Type Sample1_Beam_tutorial.rcdx in the filename and click Save.

   RCDC closes and you are returned to STAAD.Pro.

Column and Wall Design

This section of the tutorial demonstrates how to design and detail continuous concrete columns.
RCDC will make continuous physical columns out of collinear analytical columns modeled in the Building Planner mode in STAAD.Pro.

To start the column design

In STAAD.Pro, ensure you are on the Frame | Design page in the Building Planner mode.

Tip: When you import STAAD.Pro model data into RCDC, grids are automatically detected and assigned. You can manually edit both the exact locations of grids as well as grid labels in RCDC for drawing generation.

1. In the top-right dialog, click Design Columns.
RCDC opens and the **New Project** dialog opens.

2. Enter the project data:

   a. Type **Tutorial** in the **Project** field.
   
   b. (Optional) Type **Client** and **Engineer** name.
      
      This information is used as the header for all reports. It will be automatically imported from the STAAD.Pro input file Job Information block if available.
   
   c. Select **EN 02 - 2004** from the **Design Code** drop-down list.
      
      This tutorial uses the Eurocode. However, you can select any available code. Just note that some design settings and features may vary based on the code selection made here.

   The column continuity is automatically detected as the analytical model data is read.

3. Either:

   select the **Reinforcement Settings** tool 🛠️
   
   or
   
   select **Settings > Reinforcement Settings**

   The **Reinforcement Settings** dialog opens.
4. In the Rebar list, clear the check mark by both 32 (mm) and 40 (mm).
   The initial design will be evaluated using smaller size bars where possible.

5. Click OK.

Note: There are numerous other detailing and drawing settings in RCDC that allow you to customize the concrete detailing for your client or organization’s needs. For details on these options, refer to the RCDC help by pressing <F1> or selecting Help > Contents.

To import load cases and combinations

In this procedure, you will import the load settings file you saved when designing the beams (on page 4963).

1. Select Settings > Basic Load Cases.
   The Basic Load Cases dialog opens.

2. Click Import Load Cases & Combinations.
   The Open dialog opens.

3. Select the file Sample1_Loads.ldsx and then click Open.
   If you saved this file under a different name or in a different location, locate and select the file you created in the previous section.
   The previously selected load cases are added to the Primary Load Cases list.

4. Click OK.
   The Load Combinations dialog opens. The previously selected load combinations from the analysis model are also already added.
5. Click OK.

To perform the column design

1. Either:
   - select the Autodesign tool
   or
   - select Design > Autodesign.
   The Failed Columns Dialog dialog opens indicating that two columns have failed the design checks.

2. Click OK.

3. Either:
   - select the Failure Diagnostics tool
   - select Reports > Failure Diagnostics.
   The Failure Diagnostics tab opens. The table indicates that the two beams have failed a detailing check.

4. Evaluate some options for column 12:
   a. Select the row for column C12 from 6m TO 12m in the column design table.
      This column is marked with a red tag to indicate it failed one or more checks.
   b. Either:
      - select the Redesign Section tool
      - select Modify > Redesign Section
      or
      right-click on the column row and select Redesign Section from the pop-up menu.
      The Redesign Column dialog opens at the bottom of the program window.

   c. In the Dia list, select 20 (mm) and then in the Bars list select 3#.
      This will place 20mm bars in groups of three in the section.
   d. Click on any of the corner bars in the cross section.
      The color-coded size of the grouped bars are added to each corner.
e. Repeat steps 4c and 4d to add single (1#) 20mm bars along the faces.

Note: Once you select number of bars, you only need to click twice in the section to update all the face bars.

f. Select the Redesign tool in the Redesign Grp dialog. The column now passes design and detailing checks.

g. Select the Accept tool. Right-clicking on a wall and selecting Redesign Section allows you to graphically detail a wall section in the same manner. Try this with W1 in the Column Design table as an extra exercise.

The Design Output table now shows green for the G1 beam indicating it passes.

5. Repeat step 4 to redesign the other failing column (C23).
6. Right-click on the second level column 12 (C12: 6m TO 12m) and select Interaction Surface from the pop-up menu. The Interaction Surface dialog opens.
7. Review the interaction surface:
   - **Options**
     - To display the interaction at critical points for a load combination
     - To view the interaction at any point along the column height
     - To control the view of the interaction surface
   - **Description**
     - select that combination from the table
     - drag the slider to a specific load value
     - use the view controls at the top of the interaction surface view window

8. Click the X in top-right to close the Interaction Surface window.

To generate column schedules and drawings

1. Generate a text schedule of column reinforcement:
   - a. Either:
      - select the Text Schedule tool
      - or
      - Select Reports > Text Schedule > Type 1 (Ductile) Section.
      - The Text Schedule view window opens.
   - b. Click the Save tool in the view window.
   - c. Type Tutorial_Column_Sched.dxf and click Save to save the schedule as a DXF file.

2. Generate detail drawings of columns:
Application Examples
EX. Interactive Concrete Design Examples

a. Either:
   select the **Detailed Drawing** tool
   or
   Select **Reports > Detailed Drawing**.
The **Select Columns** dialog opens.
b. Select 0m to 12m for the range of heights.
c. Select C1 and W1 in the columns list.
d. Click **OK**.
The **Detailed Drawing** opens with the Column and Wall Schedule.

![Detailed Drawing Screenshot]

**e.** Click the **Save** tool in the view window.

**f.** Type `Tutorial_Col_Detail.dxf` and click **Save** to save the schedule as a DXF file.

3. Generate the elevation and sections for a column:
a. Select **Reports > Elevation**.
The **Select Columns** dialog opens.
b. Select C1 and W1 in the columns list.

   **Tip:** You can modify the drawing and detailing options by clicking the **Detailing & Drawing Settings** button (also by selecting **Settings > Detailing & Drawing Settings** before using this tool). You can also customize rebar mark styles from this dialog.

c. Click **OK**.
The elevation for the selected column and wall opens. This drawing contains section cuts at each reinforcement arrangement.
d. Click the **Save** tool in the view window.

e. Type `Tutorial_Col_Elev.dxf` and click **Save** to save the schedule as a DXF file.

4. Generate a bill of quantities:

   **a.** Either:

   select the **Bill of Quantities** tool ✡

   or

   Select **Reports > Bill of Quantities**.

   The **Bill Of Quantities** dialog opens.
b. Select the **BOQ Summary** and **Reinforcement Type Wise** options.

c. Select **€ - Euro** from the **Currency** drop-down list.

d. Type 90 (€/m³) for Grade C20/25 concrete and 4 (€/kg) for Grade Fy420 steel. Type 8 (€/m²) for the Shuttering value.

e. Click **OK**.

   The **BOQ Summary** and **BOQ Detailed - Reinf Type Wise** tabs open.

f. Either:

   click the **Print** tool to print out a copy of this report or

   click the **Save** tool to save an HTML copy of this report

5. Generate bar bending schedule:

   a. Select **BBS > Generate BBS**.

   The **Generate BBS** dialog opens.

   b. Select **C1** and **W1** in the list of columns groups.

   c. Select the **Continuous** option in the **Rebar Numbering** group.

   d. Select both the **Drawing** and **Spreadsheet** options in the **Schedule Format** group.

   e. Click inside the file path field, type **Sample1_Col_BBS** in the **Save As** dialog that opens, and then click **Save**.

   f. Click **Generate**.

   The BBS opens containing a text schedule of the bar bending schedule. If you have Microsoft® Office Excel® installed (or another spreadsheet program cable of opening .xlsx format spreadsheet files), the spreadsheet of the bar bending schedule opens.

   g. Close your spreadsheet application.

   h. Click the **Save** tool in the view window.

   i. Type **Tutorial_Col_BBS.dxf** and click **Save** to save the schedule as a DXF file.
To generate column design calculations

1. Either:
   select the Design Summary tool
   
   or
   
   select Reports > Design Summary
   
   The Design Summary opens.

2. Either:
   select the Design Calculations tool
   
   or
   
   select Reports > Design Calculations
   
   The Select Columns dialog opens.

3. Select columns C1 and W1 in the columns list.
   For this tutorial you will only output the detailed design for one column and one wall. You can check the All
   Columns option to include the full design report. Or, if you only need the calculations for one level of a
   column or wall, you can expand that entry and select the level of interest.

4. Click OK.

5. Either:
   
   click the Print tool to print out a copy of this report
   
   or
   
   click the Save tool to save an HTML copy of this report

To save the project and exit

1. Either:
   
   select the Save tool (on the main program toolbar)
   
   or
   
   select File > Save
   
   The Save As dialog opens.

2. Type Sample1_Col_tutorial.rcdx in the filename and click Save.

   
   RCDC closes and you are returned to STAAD.Pro.

Tutorial STAAD Input File

The following input file for STAAD.Pro is included with the installation of the product. It is included here as well.

```
** FILE CREATED ON : (19-09-2018, 02:49:43 PM)
** FILE IS GENERATED USING PlanWin
** PlanWin FILE : C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Building Planner\Typical Ground +3 Storey Building.plnx
UNIT METERS MTONS
JOINT COORDINATES
```
### Application Examples
#### EX. Interactive Concrete Design Examples

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>2</td>
<td>8.0000</td>
<td>0.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>3</td>
<td>16.0000</td>
<td>0.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>4</td>
<td>24.0000</td>
<td>0.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>5</td>
<td>32.0000</td>
<td>0.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>6</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>7</td>
<td>8.0000</td>
<td>0.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>8</td>
<td>16.0000</td>
<td>0.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>9</td>
<td>24.0000</td>
<td>0.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>10</td>
<td>32.0000</td>
<td>0.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>11</td>
<td>33.8700</td>
<td>0.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>12</td>
<td>46.0800</td>
<td>0.0000</td>
<td>-21.7400</td>
</tr>
<tr>
<td>13</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>14</td>
<td>8.0000</td>
<td>0.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>15</td>
<td>16.0000</td>
<td>0.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>16</td>
<td>24.0000</td>
<td>0.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>17</td>
<td>36.7900</td>
<td>0.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>18</td>
<td>37.8500</td>
<td>0.0000</td>
<td>-14.7500</td>
</tr>
<tr>
<td>19</td>
<td>49.0600</td>
<td>0.0000</td>
<td>-13.2500</td>
</tr>
<tr>
<td>20</td>
<td>0.0000</td>
<td>0.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>21</td>
<td>8.0000</td>
<td>0.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>22</td>
<td>16.0000</td>
<td>0.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>23</td>
<td>24.0000</td>
<td>0.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>24</td>
<td>32.0000</td>
<td>0.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>25</td>
<td>37.8300</td>
<td>0.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>26</td>
<td>39.7500</td>
<td>0.0000</td>
<td>-6.4600</td>
</tr>
<tr>
<td>27</td>
<td>32.0000</td>
<td>0.0000</td>
<td>-6.4600</td>
</tr>
<tr>
<td>28</td>
<td>37.8300</td>
<td>0.0000</td>
<td>-6.4600</td>
</tr>
<tr>
<td>29</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>30</td>
<td>8.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>31</td>
<td>16.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>32</td>
<td>24.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>33</td>
<td>32.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>34</td>
<td>37.8300</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>35</td>
<td>42.6800</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>36</td>
<td>32.0005</td>
<td>0.0000</td>
<td>-16.3750</td>
</tr>
<tr>
<td>37</td>
<td>33.1805</td>
<td>0.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>38</td>
<td>34.3605</td>
<td>0.0000</td>
<td>-16.3750</td>
</tr>
<tr>
<td>39</td>
<td>0.0000</td>
<td>3.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>40</td>
<td>8.0000</td>
<td>3.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>41</td>
<td>16.0000</td>
<td>3.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>42</td>
<td>24.0000</td>
<td>3.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>43</td>
<td>32.0000</td>
<td>3.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>44</td>
<td>0.0000</td>
<td>3.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>45</td>
<td>8.0000</td>
<td>3.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>46</td>
<td>16.0000</td>
<td>3.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>47</td>
<td>24.0000</td>
<td>3.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>48</td>
<td>32.0000</td>
<td>3.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>49</td>
<td>33.8700</td>
<td>3.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>50</td>
<td>46.0800</td>
<td>3.0000</td>
<td>-21.7400</td>
</tr>
<tr>
<td>51</td>
<td>0.0000</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>52</td>
<td>8.0000</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>53</td>
<td>16.0000</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>54</td>
<td>24.0000</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>55</td>
<td>32.0010</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>56</td>
<td>34.3600</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>57</td>
<td>36.7900</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>58</td>
<td>32.0000</td>
<td>3.0000</td>
<td>-14.7500</td>
</tr>
</tbody>
</table>
### Application Examples
EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Row</th>
<th>X-Value</th>
<th>Y-Value</th>
<th>Z-Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>59</td>
<td>34.3610</td>
<td>3.0000</td>
<td>-14.7500</td>
</tr>
<tr>
<td>60</td>
<td>37.8500</td>
<td>3.0000</td>
<td>-14.7500</td>
</tr>
<tr>
<td>61</td>
<td>49.0600</td>
<td>3.0000</td>
<td>-13.2500</td>
</tr>
<tr>
<td>62</td>
<td>0.0000</td>
<td>3.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>63</td>
<td>8.0000</td>
<td>3.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>64</td>
<td>16.0000</td>
<td>3.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>65</td>
<td>24.0000</td>
<td>3.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>66</td>
<td>32.0000</td>
<td>3.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>67</td>
<td>37.8300</td>
<td>3.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>68</td>
<td>39.7500</td>
<td>3.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>69</td>
<td>32.0000</td>
<td>3.0000</td>
<td>-6.4600</td>
</tr>
<tr>
<td>70</td>
<td>37.8300</td>
<td>3.0000</td>
<td>-6.4600</td>
</tr>
<tr>
<td>71</td>
<td>0.0000</td>
<td>3.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>72</td>
<td>8.0000</td>
<td>3.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>73</td>
<td>16.0000</td>
<td>3.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>74</td>
<td>24.0000</td>
<td>3.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>75</td>
<td>32.0000</td>
<td>3.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>76</td>
<td>37.8300</td>
<td>3.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>77</td>
<td>42.6800</td>
<td>3.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>78</td>
<td>32.0005</td>
<td>3.0000</td>
<td>-16.3750</td>
</tr>
<tr>
<td>79</td>
<td>33.1805</td>
<td>3.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>80</td>
<td>34.3605</td>
<td>3.0000</td>
<td>-16.3750</td>
</tr>
<tr>
<td>81</td>
<td>0.0000</td>
<td>6.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>82</td>
<td>8.0000</td>
<td>6.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>83</td>
<td>16.0000</td>
<td>6.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>84</td>
<td>24.0000</td>
<td>6.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>85</td>
<td>32.0000</td>
<td>6.0000</td>
<td>-32.6100</td>
</tr>
<tr>
<td>86</td>
<td>0.0000</td>
<td>6.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>87</td>
<td>8.0000</td>
<td>6.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>88</td>
<td>16.0000</td>
<td>6.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>89</td>
<td>24.0000</td>
<td>6.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>90</td>
<td>32.0000</td>
<td>6.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>91</td>
<td>33.8700</td>
<td>6.0000</td>
<td>-26.9000</td>
</tr>
<tr>
<td>92</td>
<td>46.0800</td>
<td>6.0000</td>
<td>-21.7400</td>
</tr>
<tr>
<td>93</td>
<td>0.0000</td>
<td>6.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>94</td>
<td>8.0000</td>
<td>6.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>95</td>
<td>16.0000</td>
<td>6.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>96</td>
<td>24.0000</td>
<td>6.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>97</td>
<td>32.0010</td>
<td>6.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>98</td>
<td>34.3600</td>
<td>6.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>99</td>
<td>36.7900</td>
<td>6.0000</td>
<td>-18.0000</td>
</tr>
<tr>
<td>100</td>
<td>32.0000</td>
<td>6.0000</td>
<td>-14.7500</td>
</tr>
<tr>
<td>101</td>
<td>34.3610</td>
<td>6.0000</td>
<td>-14.7500</td>
</tr>
<tr>
<td>102</td>
<td>37.8500</td>
<td>6.0000</td>
<td>-14.7500</td>
</tr>
<tr>
<td>103</td>
<td>49.0600</td>
<td>6.0000</td>
<td>-13.2500</td>
</tr>
<tr>
<td>104</td>
<td>0.0000</td>
<td>6.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>105</td>
<td>8.0000</td>
<td>6.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>106</td>
<td>16.0000</td>
<td>6.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>107</td>
<td>24.0000</td>
<td>6.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>108</td>
<td>32.0000</td>
<td>6.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>109</td>
<td>33.8700</td>
<td>6.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>110</td>
<td>46.0800</td>
<td>6.0000</td>
<td>-8.9500</td>
</tr>
<tr>
<td>111</td>
<td>32.0000</td>
<td>6.0000</td>
<td>-6.4600</td>
</tr>
<tr>
<td>112</td>
<td>37.8300</td>
<td>6.0000</td>
<td>-6.4600</td>
</tr>
<tr>
<td>113</td>
<td>0.0000</td>
<td>6.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>114</td>
<td>8.0000</td>
<td>6.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>115</td>
<td>16.0000</td>
<td>6.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>116</td>
<td>24.0000</td>
<td>6.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
**Application Examples**

**EX. Interactive Concrete Design Examples**

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>117</td>
<td>32.000</td>
<td>6.000</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>118</td>
<td>37.830</td>
<td>6.000</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>119</td>
<td>42.680</td>
<td>6.000</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>120</td>
<td>32.0005</td>
<td>6.000</td>
<td>-16.375</td>
<td></td>
</tr>
<tr>
<td>121</td>
<td>33.1805</td>
<td>6.000</td>
<td>-18.000</td>
<td></td>
</tr>
<tr>
<td>122</td>
<td>34.3605</td>
<td>6.000</td>
<td>-16.375</td>
<td></td>
</tr>
<tr>
<td>123</td>
<td>0.0000</td>
<td>9.0000</td>
<td>-32.6100</td>
<td></td>
</tr>
<tr>
<td>124</td>
<td>8.0000</td>
<td>9.0000</td>
<td>-32.6100</td>
<td></td>
</tr>
<tr>
<td>125</td>
<td>16.0000</td>
<td>9.0000</td>
<td>-32.6100</td>
<td></td>
</tr>
<tr>
<td>126</td>
<td>24.0000</td>
<td>9.0000</td>
<td>-32.6100</td>
<td></td>
</tr>
<tr>
<td>127</td>
<td>32.0000</td>
<td>9.0000</td>
<td>-32.6100</td>
<td></td>
</tr>
<tr>
<td>128</td>
<td>0.0000</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>129</td>
<td>8.0000</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>130</td>
<td>16.0000</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>131</td>
<td>24.0000</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>132</td>
<td>32.0000</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>133</td>
<td>33.1805</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>134</td>
<td>34.3605</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>135</td>
<td>36.7900</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>136</td>
<td>38.9700</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>137</td>
<td>41.2100</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>138</td>
<td>43.4500</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>139</td>
<td>45.7900</td>
<td>9.0000</td>
<td>-26.9000</td>
<td></td>
</tr>
<tr>
<td>140</td>
<td>33.1805</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>141</td>
<td>34.3605</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>142</td>
<td>36.7900</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>143</td>
<td>39.2200</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>144</td>
<td>41.6500</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>145</td>
<td>44.0800</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>146</td>
<td>46.5100</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>147</td>
<td>48.9400</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>148</td>
<td>51.3700</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>149</td>
<td>53.8000</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>150</td>
<td>56.2300</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>151</td>
<td>58.6600</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>152</td>
<td>61.0900</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>153</td>
<td>63.5200</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>154</td>
<td>65.9500</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>155</td>
<td>68.3800</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>156</td>
<td>70.8100</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>157</td>
<td>73.2400</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>158</td>
<td>75.6700</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>159</td>
<td>78.1000</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>160</td>
<td>80.5300</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>161</td>
<td>82.9600</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>162</td>
<td>85.3900</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>163</td>
<td>87.8200</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>164</td>
<td>90.2500</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>165</td>
<td>92.6800</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>166</td>
<td>95.1100</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>167</td>
<td>97.5400</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>168</td>
<td>99.9700</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>169</td>
<td>102.4000</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>170</td>
<td>104.8300</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>171</td>
<td>107.2600</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>172</td>
<td>109.6900</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>173</td>
<td>112.1200</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
<tr>
<td>174</td>
<td>114.5500</td>
<td>9.0000</td>
<td>-21.7400</td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>Column 1</th>
<th>Column 2</th>
<th>Column 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>175</td>
<td>33.8700</td>
<td>12.0000</td>
</tr>
<tr>
<td>176</td>
<td>0.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>177</td>
<td>8.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>178</td>
<td>16.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>179</td>
<td>24.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>180</td>
<td>32.0010</td>
<td>12.0000</td>
</tr>
<tr>
<td>181</td>
<td>34.3600</td>
<td>12.0000</td>
</tr>
<tr>
<td>182</td>
<td>36.7900</td>
<td>12.0000</td>
</tr>
<tr>
<td>183</td>
<td>32.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>184</td>
<td>34.3610</td>
<td>12.0000</td>
</tr>
<tr>
<td>185</td>
<td>37.8500</td>
<td>12.0000</td>
</tr>
<tr>
<td>186</td>
<td>0.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>187</td>
<td>8.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>188</td>
<td>16.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>189</td>
<td>24.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>190</td>
<td>32.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>191</td>
<td>37.8300</td>
<td>12.0000</td>
</tr>
<tr>
<td>192</td>
<td>39.7500</td>
<td>12.0000</td>
</tr>
<tr>
<td>193</td>
<td>32.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>194</td>
<td>37.8300</td>
<td>12.0000</td>
</tr>
<tr>
<td>195</td>
<td>0.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>196</td>
<td>8.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>197</td>
<td>16.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>198</td>
<td>24.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>199</td>
<td>32.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>200</td>
<td>37.8300</td>
<td>12.0000</td>
</tr>
<tr>
<td>201</td>
<td>42.6800</td>
<td>12.0000</td>
</tr>
<tr>
<td>202</td>
<td>32.0005</td>
<td>12.0000</td>
</tr>
<tr>
<td>203</td>
<td>33.1805</td>
<td>12.0000</td>
</tr>
<tr>
<td>204</td>
<td>34.3605</td>
<td>12.0000</td>
</tr>
</tbody>
</table>

**MEMBER INCIDENCES**

*Columns at Level 1*

1 1  39  
2 2  40  
3 3  41  
4 4  42  
5 5  43  
6 6  44  
7 7  45  
8 8  46  
9 9  47  
10 10  48  
11 11  49  
12 12  50  
13 13  51  
14 14  52  
15 15  53  
16 16  54  
17 17  57  
18 18  60  
19 19  61  
20 20  62  
21 21  63  
22 22  64  
23 23  65  
24 24  66  
25 25  67  
26 26  68  

STAAD.Pro 4983 User Manual
Application Examples
EX. Interactive Concrete Design Examples

| 27 27 69 |
| 28 28 70 |
| 29 29 71 |
| 30 30 72 |
| 31 31 73 |
| 32 32 74 |
| 33 33 75 |
| 34 34 76 |
| 35 35 77 |
| 36 36 78 |
| 37 37 79 |
| 38 38 80 |

*Columns at Level 2
| 39 39 81 |
| 40 40 82 |
| 41 41 83 |
| 42 42 84 |
| 43 43 85 |
| 44 44 86 |
| 45 45 87 |
| 46 46 88 |
| 47 47 89 |
| 48 48 90 |
| 49 49 91 |
| 50 50 92 |
| 51 51 93 |
| 52 52 94 |
| 53 53 95 |
| 54 54 96 |
| 55 57 99 |
| 56 60 102 |
| 57 61 103 |
| 58 62 104 |
| 59 63 105 |
| 60 64 106 |
| 61 65 107 |
| 62 66 108 |
| 63 67 109 |
| 64 68 110 |
| 65 69 111 |
| 66 70 112 |
| 67 71 113 |
| 68 72 114 |
| 69 73 115 |
| 70 74 116 |
| 71 75 117 |
| 72 76 118 |
| 73 77 119 |
| 74 78 120 |
| 75 79 121 |
| 76 80 122 |

*Columns at Level 3
| 77 81 123 |
| 78 82 124 |
| 79 83 125 |
| 80 84 126 |
| 81 85 127 |
| 82 86 128 |
Application Examples
EX. Interactive Concrete Design Examples
Application Examples

EX. Interactive Concrete Design Examples

Beams at Level 1

151 39  40
152 40  41
153 41  42
154 42  43
155 44  45
156 45  46
157 46  47
158 47  48
159 48  49
160 51  52
161 52  53
162 53  54
163 54  55
164 55  79
165 56  57
166 58  59
167 59  60
168 62  63
169 63  64
170 64  65
171 65  66
172 66  67
173 67  68
174 69  70
175 71  72
176 72  73
177 73  74
178 74  75
179 75  76
180 76  77
181 71  62
182 62  51
183 51  44
184 44  39
185 73  64
186 63  52
187 52  45
188 45  40
189 73  64
Application Examples
EX. Interactive Concrete Design Examples

*Beams at Level 2

217 81  82
218 82  83
219 83  84
220 84  85
221 86  87
222 87  88
223 88  89
224 89  90
225 90  91
226 93  94
227 94  95
228 95  96
229 96  97
230 97  121
231 98  99
232 100 101
233 101 102
234 104 105
235 105 106
236 106 107
237 107 108
238 108 109
239 109 110
240 111 112
241 113 114
242 114 115
243 115 116
244 116 117
245 117 118
246 118 119
247 113 104
248 104  93
249 93   86
250 86   81
251 114 105
252 105  94
253 94   87
### Application Examples

EX. Interactive Concrete Design Examples

*Beams at Level 3*

<table>
<thead>
<tr>
<th>283</th>
<th>123</th>
<th>124</th>
</tr>
</thead>
<tbody>
<tr>
<td>284</td>
<td>124</td>
<td>125</td>
</tr>
<tr>
<td>285</td>
<td>125</td>
<td>126</td>
</tr>
<tr>
<td>286</td>
<td>126</td>
<td>127</td>
</tr>
<tr>
<td>287</td>
<td>128</td>
<td>129</td>
</tr>
<tr>
<td>288</td>
<td>129</td>
<td>130</td>
</tr>
<tr>
<td>289</td>
<td>130</td>
<td>131</td>
</tr>
<tr>
<td>290</td>
<td>131</td>
<td>132</td>
</tr>
<tr>
<td>291</td>
<td>132</td>
<td>133</td>
</tr>
<tr>
<td>292</td>
<td>135</td>
<td>136</td>
</tr>
<tr>
<td>293</td>
<td>136</td>
<td>137</td>
</tr>
<tr>
<td>294</td>
<td>137</td>
<td>138</td>
</tr>
<tr>
<td>295</td>
<td>138</td>
<td>139</td>
</tr>
<tr>
<td>296</td>
<td>139</td>
<td>143</td>
</tr>
<tr>
<td>297</td>
<td>140</td>
<td>141</td>
</tr>
<tr>
<td>298</td>
<td>142</td>
<td>143</td>
</tr>
<tr>
<td>299</td>
<td>143</td>
<td>144</td>
</tr>
<tr>
<td>300</td>
<td>146</td>
<td>147</td>
</tr>
<tr>
<td>301</td>
<td>147</td>
<td>148</td>
</tr>
<tr>
<td>302</td>
<td>148</td>
<td>149</td>
</tr>
<tr>
<td>303</td>
<td>149</td>
<td>150</td>
</tr>
<tr>
<td>304</td>
<td>150</td>
<td>151</td>
</tr>
<tr>
<td>305</td>
<td>151</td>
<td>152</td>
</tr>
<tr>
<td>306</td>
<td>153</td>
<td>154</td>
</tr>
<tr>
<td>307</td>
<td>155</td>
<td>156</td>
</tr>
<tr>
<td>308</td>
<td>156</td>
<td>157</td>
</tr>
<tr>
<td>309</td>
<td>157</td>
<td>158</td>
</tr>
<tr>
<td>310</td>
<td>158</td>
<td>159</td>
</tr>
</tbody>
</table>
Application Examples
EX. Interactive Concrete Design Examples

*Beams at Level 4

STAAD.Pro 4989 User Manual
Application Examples
EX. Interactive Concrete Design Examples

```plaintext
DEFINE MATERIAL START
  ISOTROPIC M20
    E 2236067.97749979
    POISSON 0.17
    DENSITY 2.5
    ALPHA 1e-005
    DAMP 0.05
  ISOTROPIC RL20
    E 2236067.97749979
    POISSON 0.17
    DENSITY 0
    ALPHA 1e-005
    DAMP 0.05
END DEFINE MATERIAL
```
### MEMBER PROPERTIES

*Columns*

<table>
<thead>
<tr>
<th>Member</th>
<th>PRIS</th>
<th>YD</th>
<th>ZD</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 5 35 39 43 73</td>
<td>0.23</td>
<td>0.3</td>
<td></td>
</tr>
<tr>
<td>2 4 33 40 42 71 103 139</td>
<td>0.23</td>
<td>0.7</td>
<td></td>
</tr>
<tr>
<td>3 41 95 104 140</td>
<td>0.23</td>
<td>0.55</td>
<td></td>
</tr>
<tr>
<td>6 44 107 143</td>
<td>0.23</td>
<td>0.85</td>
<td></td>
</tr>
<tr>
<td>7 TO 9 14 TO 16 21 TO 23 45 TO 47 52 TO 54 59 TO 61</td>
<td>0.23</td>
<td>0.6</td>
<td></td>
</tr>
<tr>
<td>10 48</td>
<td>0.23</td>
<td>1.3</td>
<td></td>
</tr>
<tr>
<td>11 25 49 63 81 87 101 111 115 119 125 137 147</td>
<td>0.23</td>
<td>0.25</td>
<td></td>
</tr>
<tr>
<td>12 50 78 80 109 116 118 145</td>
<td>0.23</td>
<td>0.6</td>
<td></td>
</tr>
<tr>
<td>13 32 51 70 96 132</td>
<td>0.23</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>17 55 82 120</td>
<td>0.23</td>
<td>0.75</td>
<td></td>
</tr>
<tr>
<td>18 56 79 117</td>
<td>0.23</td>
<td>0.45</td>
<td></td>
</tr>
<tr>
<td>19 28 57 66 93 130</td>
<td>0.23</td>
<td>0.65</td>
<td></td>
</tr>
<tr>
<td>20 58</td>
<td>0.23</td>
<td>1.1</td>
<td></td>
</tr>
<tr>
<td>24 26 62 64 89 108 126 144</td>
<td>0.23</td>
<td>0.9</td>
<td></td>
</tr>
<tr>
<td>27 65 100 102 136 138</td>
<td>0.23</td>
<td>0.8</td>
<td></td>
</tr>
<tr>
<td>29 34 67 72 88</td>
<td>0.23</td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td>30 68</td>
<td>0.23</td>
<td>1.05</td>
<td></td>
</tr>
<tr>
<td>31 69 106 142</td>
<td>0.23</td>
<td>0.95</td>
<td></td>
</tr>
<tr>
<td>36 38 74 76 112 114 148 150</td>
<td>0.23</td>
<td>3.25000015367718</td>
<td></td>
</tr>
<tr>
<td>37 75 113 149</td>
<td>0.23</td>
<td>2.35900115966797</td>
<td></td>
</tr>
<tr>
<td>83 TO 85 90 TO 92 97 TO 99</td>
<td>0.55</td>
<td>0.55</td>
<td></td>
</tr>
<tr>
<td>86 124</td>
<td>0.23</td>
<td>1.2</td>
<td></td>
</tr>
<tr>
<td>94 105 110 131 141 146</td>
<td>0.23</td>
<td>0.4</td>
<td></td>
</tr>
<tr>
<td>121 TO 123 127 TO 129 132 TO 135</td>
<td>0.23</td>
<td>0.5</td>
<td></td>
</tr>
</tbody>
</table>

*Beams*

<table>
<thead>
<tr>
<th>Member</th>
<th>PRIS</th>
<th>YD</th>
<th>ZD</th>
</tr>
</thead>
<tbody>
<tr>
<td>151 TO 163 165 TO 199</td>
<td>0.23</td>
<td>0.23</td>
<td></td>
</tr>
<tr>
<td>201 TO 202 204 TO 213 217 TO 229</td>
<td>0.23</td>
<td>0.75</td>
<td></td>
</tr>
<tr>
<td>402 TO 408</td>
<td>0.75</td>
<td>0.23</td>
<td></td>
</tr>
<tr>
<td>164 200 203 214 TO 216 230 266 269 280 TO 282 296 332</td>
<td>0.23</td>
<td>0.23</td>
<td></td>
</tr>
</tbody>
</table>

### MEMBER RELEASE

**CONSTANTS**

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>M20</th>
<th>MEMB 1 TO 150</th>
</tr>
</thead>
<tbody>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 151 TO 163</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 165 TO 199</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 201 TO 202</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 204 TO 213</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 217 TO 229</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 231 TO 265</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 267 TO 268</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 270 TO 279</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 283 TO 295</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 297 TO 331</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 333 TO 334</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 336 TO 345</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 349 TO 361</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 363 TO 397</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 399 TO 400</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>M20</td>
<td>MEMB 402 TO 408</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>RL20</td>
<td>MEMB 164</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>RL20</td>
<td>MEMB 200</td>
</tr>
<tr>
<td>MATERIAL</td>
<td>RL20</td>
<td>MEMB 203</td>
</tr>
</tbody>
</table>
Application Examples
EX. Interactive Concrete Design Examples

MATERIAL RL20 MEMB 214 TO 216
MATERIAL RL20 MEMB 230
MATERIAL RL20 MEMB 266
MATERIAL RL20 MEMB 269
MATERIAL RL20 MEMB 280 TO 282
MATERIAL RL20 MEMB 296
MATERIAL RL20 MEMB 332
MATERIAL RL20 MEMB 335
MATERIAL RL20 MEMB 346 TO 348
MATERIAL RL20 MEMB 362
MATERIAL RL20 MEMB 398
MATERIAL RL20 MEMB 401
MATERIAL RL20 MEMB 409 TO 411
*Columns Beta Angle
BETA 270 MEM 1
BETA 180 MEM 2 TO 4
BETA 198 MEM 5
BETA 270 MEM 6 TO 10
BETA 198 MEM 11
BETA 112 MEM 12
BETA 270 MEM 13 TO 16
BETA 18 MEM 17 TO 18
BETA 115 MEM 19
BETA 270 MEM 20 TO 25
BETA 18 MEM 26
BETA 270 MEM 27 TO 29
BETA 18 MEM 35
BETA 270 MEM 37
BETA 180 MEM 38
BETA 270 MEM 39
BETA 180 MEM 40 TO 42
BETA 198 MEM 43
BETA 270 MEM 44 TO 48
BETA 198 MEM 49
BETA 112 MEM 50
BETA 270 MEM 51 TO 54
BETA 18 MEM 55 TO 56
BETA 115 MEM 57
BETA 270 MEM 58 TO 63
BETA 18 MEM 64
BETA 270 MEM 65 TO 67
BETA 18 MEM 73
BETA 270 MEM 75
BETA 180 MEM 76
BETA 270 MEM 77
BETA 180 MEM 78 TO 80
BETA 198 MEM 81
BETA 270 MEM 82 TO 86
BETA 198 MEM 87
BETA 112 MEM 88
BETA 270 MEM 89 TO 92
BETA 18 MEM 93 TO 94
BETA 115 MEM 95
BETA 270 MEM 96 TO 101
BETA 18 MEM 102
BETA 270 MEM 103 TO 105
BETA 18 MEM 111
BETA 270 MEM 113
Application Examples
EX. Interactive Concrete Design Examples

BETA 180 MEM 114
BETA 270 MEM 115
BETA 180 MEM 116 TO 118
BETA 198 MEM 119
BETA 270 MEM 120 TO 124
BETA 198 MEM 125
BETA 270 MEM 126 TO 129
BETA 18 MEM 130 TO 131
BETA 270 MEM 132 TO 137
BETA 18 MEM 138
BETA 270 MEM 139 TO 141
BETA 18 MEM 147
BETA 270 MEM 149
BETA 180 MEM 150
*All Beam Beta Angle=0
BETA 0 MEM 151 TO 411

SUPPORTS
1 FIXED
2 FIXED
3 FIXED
4 FIXED
5 FIXED
6 FIXED
7 FIXED
8 FIXED
9 FIXED
10 FIXED
11 FIXED
12 FIXED
13 FIXED
14 FIXED
15 FIXED
16 FIXED
17 FIXED
18 FIXED
19 FIXED
20 FIXED
21 FIXED
22 FIXED
23 FIXED
24 FIXED
25 FIXED
26 FIXED
27 FIXED
28 FIXED
29 FIXED
30 FIXED
31 FIXED
32 FIXED
33 FIXED
34 FIXED
35 FIXED
36 FIXED
37 FIXED
38 FIXED

LOAD 5 LOADTYPE Wind TITLE WIND IN X-DIR
JOINT LOAD
39 FX 5.768697E-02
Application Examples
EX. Interactive Concrete Design Examples

1 FX 5.768697E-02
40 FX 5.768697E-02
2 FX 5.768697E-02
41 FX 5.768697E-02
3 FX 5.768697E-02
42 FX 5.768697E-02
4 FX 5.768697E-02
43 FX 5.768697E-02
5 FX 5.768697E-02
44 FX 9.151451E-02
6 FX 9.151451E-02
45 FX 9.151451E-02
7 FX 9.151451E-02
46 FX 9.151451E-02
8 FX 9.151451E-02
47 FX 9.151451E-02
9 FX 9.151451E-02
48 FX 9.151451E-02
10 FX 9.151451E-02
49 FX 9.151451E-02
11 FX 9.151451E-02
50 FX 0.4495745
12 FX 0.4495745
51 FX 5.044183E-02
13 FX 5.044183E-02
52 FX 5.044183E-02
14 FX 5.044183E-02
53 FX 5.044183E-02
15 FX 5.044183E-02
54 FX 5.044183E-02
16 FX 5.044183E-02
57 FX 5.044183E-02
17 FX 5.044183E-02
60 FX 7.998049E-02
18 FX 7.998049E-02
61 FX 0.2929811
19 FX 0.2929811
62 FX 4.899858E-02
20 FX 4.899858E-02
63 FX 4.899858E-02
21 FX 4.899858E-02
64 FX 4.899858E-02
22 FX 4.899858E-02
65 FX 4.899858E-02
23 FX 4.899858E-02
66 FX 4.899858E-02
24 FX 4.899858E-02
67 FX 4.899858E-02
25 FX 4.899858E-02
68 FX 4.899858E-02
26 FX 4.899858E-02
69 FX 0.2260501
27 FX 0.2260501
70 FX 0.2260501
28 FX 0.2260501
71 FX 0.0466172
29 FX 0.0466172
72 FX 0.0466172
Application Examples
EX. Interactive Concrete Design Examples
<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>63</td>
<td>FX 7.349786E-02</td>
</tr>
<tr>
<td>64</td>
<td>FX 7.349786E-02</td>
</tr>
<tr>
<td>65</td>
<td>FX 7.349786E-02</td>
</tr>
<tr>
<td>66</td>
<td>FX 7.349786E-02</td>
</tr>
<tr>
<td>67</td>
<td>FX 7.349786E-02</td>
</tr>
<tr>
<td>68</td>
<td>FX 7.349786E-02</td>
</tr>
<tr>
<td>69</td>
<td>FX 0.3390752</td>
</tr>
<tr>
<td>70</td>
<td>FX 0.3390752</td>
</tr>
<tr>
<td>71</td>
<td>FX 6.992579E-02</td>
</tr>
<tr>
<td>72</td>
<td>FX 6.992579E-02</td>
</tr>
<tr>
<td>73</td>
<td>FX 6.992579E-02</td>
</tr>
<tr>
<td>74</td>
<td>FX 6.992579E-02</td>
</tr>
<tr>
<td>75</td>
<td>FX 6.992579E-02</td>
</tr>
<tr>
<td>76</td>
<td>FX 6.992579E-02</td>
</tr>
<tr>
<td>77</td>
<td>FX 6.992579E-02</td>
</tr>
<tr>
<td>78</td>
<td>FX 0.1199707</td>
</tr>
<tr>
<td>79</td>
<td>FX 0.1199707</td>
</tr>
<tr>
<td>80</td>
<td>FX 7.566273E-02</td>
</tr>
<tr>
<td>81</td>
<td>FX 8.653045E-02</td>
</tr>
<tr>
<td>82</td>
<td>FX 8.653045E-02</td>
</tr>
<tr>
<td>83</td>
<td>FX 8.653045E-02</td>
</tr>
<tr>
<td>84</td>
<td>FX 8.653045E-02</td>
</tr>
<tr>
<td>85</td>
<td>FX 8.653045E-02</td>
</tr>
<tr>
<td>86</td>
<td>FX 0.1372718</td>
</tr>
<tr>
<td>87</td>
<td>FX 0.1372718</td>
</tr>
<tr>
<td>88</td>
<td>FX 0.1372718</td>
</tr>
<tr>
<td>89</td>
<td>FX 0.1372718</td>
</tr>
<tr>
<td>90</td>
<td>FX 0.1372718</td>
</tr>
<tr>
<td>91</td>
<td>FX 0.1372718</td>
</tr>
<tr>
<td>92</td>
<td>FX 0.6743617</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>92</th>
<th>FX</th>
<th>0.6743617</th>
</tr>
</thead>
<tbody>
<tr>
<td>135</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>93</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>136</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>94</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>137</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>95</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>138</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>96</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>141</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>99</td>
<td>FX</td>
<td>7.566274E-02</td>
</tr>
<tr>
<td>144</td>
<td>FX</td>
<td>0.1199707</td>
</tr>
<tr>
<td>102</td>
<td>FX</td>
<td>0.1199707</td>
</tr>
<tr>
<td>145</td>
<td>FX</td>
<td>0.4394716</td>
</tr>
<tr>
<td>103</td>
<td>FX</td>
<td>0.4394716</td>
</tr>
<tr>
<td>146</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>104</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>147</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>105</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>148</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>106</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>149</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>107</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>150</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>108</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>151</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>109</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>152</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>110</td>
<td>FX</td>
<td>7.349786E-02</td>
</tr>
<tr>
<td>153</td>
<td>FX</td>
<td>0.3390752</td>
</tr>
<tr>
<td>111</td>
<td>FX</td>
<td>0.3390752</td>
</tr>
<tr>
<td>154</td>
<td>FX</td>
<td>0.3390752</td>
</tr>
<tr>
<td>112</td>
<td>FX</td>
<td>0.3390752</td>
</tr>
<tr>
<td>155</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>113</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>156</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>114</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>157</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>115</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>158</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>116</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>159</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>117</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>160</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>118</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>161</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>119</td>
<td>FX</td>
<td>6.992579E-02</td>
</tr>
<tr>
<td>162</td>
<td>FX</td>
<td>0.1199707</td>
</tr>
<tr>
<td>162</td>
<td>FX</td>
<td>0.1199707</td>
</tr>
<tr>
<td>163</td>
<td>FX</td>
<td>7.566273E-02</td>
</tr>
<tr>
<td>163</td>
<td>FX</td>
<td>7.566273E-02</td>
</tr>
<tr>
<td>164</td>
<td>FX</td>
<td>0.1956334</td>
</tr>
<tr>
<td>164</td>
<td>FX</td>
<td>0.1956334</td>
</tr>
<tr>
<td>165</td>
<td>FX</td>
<td>9.330964E-02</td>
</tr>
<tr>
<td>123</td>
<td>FX</td>
<td>0.0878863</td>
</tr>
<tr>
<td>166</td>
<td>FX</td>
<td>9.330964E-02</td>
</tr>
<tr>
<td>124</td>
<td>FX</td>
<td>0.0878863</td>
</tr>
<tr>
<td>167</td>
<td>FX</td>
<td>9.330964E-02</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

| 125 FX | 0.0878863 |
| 126 FX | 0.0878863 |
| 127 FX | 0.0878863 |
| 128 FX | 0.1873933 |
| 129 FX | 0.1873933 |
| 130 FX | 0.1873933 |
| 131 FX | 0.1873933 |
| 132 FX | 0.1873933 |
| 133 FX | 0.1873933 |
| 134 FX | 0.1873933 |
| 135 FX | 0.1873933 |
| 136 FX | 0.1873933 |
| 137 FX | 0.1873933 |
| 138 FX | 0.1873933 |
| 139 FX | 0.1873933 |
| 140 FX | 0.1873933 |
| 141 FX | 0.1873933 |
| 142 FX | 0.1873933 |
| 143 FX | 0.1873933 |
| 144 FX | 0.1873933 |
| 145 FX | 0.1873933 |
| 146 FX | 0.1873933 |
| 147 FX | 0.1873933 |
| 148 FX | 0.1873933 |
| 149 FX | 0.1873933 |
| 150 FX | 0.1873933 |
| 151 FX | 0.1873933 |
| 152 FX | 0.1873933 |
| 153 FX | 0.1873933 |
| 154 FX | 0.1873933 |
| 155 FX | 0.1873933 |
| 156 FX | 0.1873933 |
| 157 FX | 0.1873933 |
| 158 FX | 0.1873933 |
| 159 FX | 0.1873933 |
| 160 FX | 0.1873933 |
| 161 FX | 0.1873933 |
| 162 FX | 0.1873933 |
| 163 FX | 0.1873933 |
| 164 FX | 0.1873933 |
| 165 FX | 0.1873933 |
| 166 FX | 0.1873933 |
| 167 FX | 0.1873933 |
| 168 FX | 0.1873933 |
| 169 FX | 0.1873933 |
| 170 FX | 0.1873933 |
| 171 FX | 0.1873933 |
| 172 FX | 0.1873933 |
| 173 FX | 0.1873933 |
| 174 FX | 0.1873933 |
| 175 FX | 0.1873933 |
| 176 FX | 0.1873933 |
| 177 FX | 0.1873933 |
| 178 FX | 0.1873933 |
| 179 FX | 0.1873933 |
| 180 FX | 0.1873933 |
| 181 FX | 0.1873933 |
| 182 FX | 0.1873933 |
| 183 FX | 0.1873933 |
| 184 FX | 0.1873933 |
| 185 FX | 0.1873933 |
| 186 FX | 0.1873933 |
| 187 FX | 0.1873933 |
| 188 FX | 0.1873933 |
| 189 FX | 0.1873933 |
| 190 FX | 0.1873933 |
| 191 FX | 0.1873933 |
| 192 FX | 0.1873933 |
| 193 FX | 0.1873933 |
| 194 FX | 0.1873933 |
| 195 FX | 0.1873933 |
| 196 FX | 0.1873933 |
| 197 FX | 0.1873933 |
| 198 FX | 0.1873933 |
| 199 FX | 0.1873933 |
| 200 FX | 0.1873933 |
# Application Examples

EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Load No</th>
<th>FX</th>
<th>Force</th>
<th>Type</th>
<th>Joint Load</th>
<th>Title</th>
<th>Wind in Z-Dir</th>
</tr>
</thead>
<tbody>
<tr>
<td>160</td>
<td>7.102145E-02</td>
<td>201</td>
<td>FX</td>
<td>7.540409E-02</td>
<td>161</td>
<td>7.102145E-02</td>
</tr>
<tr>
<td>202</td>
<td>0.2464834</td>
<td>203</td>
<td>FX</td>
<td>0.1418204</td>
<td>204</td>
<td>0.3883038</td>
</tr>
<tr>
<td>204</td>
<td>0.3657348</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>LOAD 6 LOADTYPE Wind TITLE WIND IN Z-DIR</td>
<td>JOINT LOAD</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>-8.758195E-02</td>
<td>1</td>
<td>FZ</td>
<td>-8.758195E-02</td>
<td>40</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>40</td>
<td>-0.1751639</td>
<td>41</td>
<td>FZ</td>
<td>-0.1751639</td>
<td>42</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>42</td>
<td>-0.1751639</td>
<td>43</td>
<td>FZ</td>
<td>-0.0771816</td>
<td>44</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>44</td>
<td>-0.0771816</td>
<td>45</td>
<td>FZ</td>
<td>-0.1751639</td>
<td>46</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>46</td>
<td>-0.1751639</td>
<td>47</td>
<td>FZ</td>
<td>-0.1751639</td>
<td>48</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>48</td>
<td>-0.0771816</td>
<td>49</td>
<td>FZ</td>
<td>-0.1291834</td>
<td>50</td>
<td>-0.3492331</td>
</tr>
<tr>
<td>50</td>
<td>-0.3492331</td>
<td>51</td>
<td>FZ</td>
<td>-8.758195E-02</td>
<td>52</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>52</td>
<td>-0.1751639</td>
<td>53</td>
<td>FZ</td>
<td>-0.1751639</td>
<td>54</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>54</td>
<td>-0.1751639</td>
<td>55</td>
<td>FZ</td>
<td>-0.1751639</td>
<td>56</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>56</td>
<td>-0.0771816</td>
<td>57</td>
<td>FZ</td>
<td>-0.1291834</td>
<td>58</td>
<td>-0.3492331</td>
</tr>
<tr>
<td>58</td>
<td>-0.3492331</td>
<td>59</td>
<td>FZ</td>
<td>-8.758195E-02</td>
<td>60</td>
<td>-0.1631214</td>
</tr>
<tr>
<td>60</td>
<td>-0.1631214</td>
<td>61</td>
<td>FZ</td>
<td>-0.1631214</td>
<td>62</td>
<td>-8.758195E-02</td>
</tr>
<tr>
<td>62</td>
<td>-8.758195E-02</td>
<td>63</td>
<td>FZ</td>
<td>-0.1751639</td>
<td>64</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>64</td>
<td>-0.1751639</td>
<td>65</td>
<td>FZ</td>
<td>-0.1751639</td>
<td>66</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>66</td>
<td>-0.0771816</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>Position</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>24 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>67 FZ</td>
<td>-0.050666E-02</td>
</tr>
<tr>
<td>25 FZ</td>
<td>-0.050666E-02</td>
</tr>
<tr>
<td>68 FZ</td>
<td>-0.2654828</td>
</tr>
<tr>
<td>26 FZ</td>
<td>-0.2654828</td>
</tr>
<tr>
<td>69 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>27 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>70 FZ</td>
<td>-0.050666E-02</td>
</tr>
<tr>
<td>28 FZ</td>
<td>-0.050666E-02</td>
</tr>
<tr>
<td>71 FZ</td>
<td>-0.758195E-02</td>
</tr>
<tr>
<td>29 FZ</td>
<td>-0.758195E-02</td>
</tr>
<tr>
<td>72 FZ</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>30 FZ</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>73 FZ</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>31 FZ</td>
<td>-0.1751639</td>
</tr>
<tr>
<td>74 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>32 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>75 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>33 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>76 FZ</td>
<td>-0.050666E-02</td>
</tr>
<tr>
<td>34 FZ</td>
<td>-0.050666E-02</td>
</tr>
<tr>
<td>77 FZ</td>
<td>-0.3464961</td>
</tr>
<tr>
<td>35 FZ</td>
<td>-0.3464961</td>
</tr>
<tr>
<td>78 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>36 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>79 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>37 FZ</td>
<td>-0.0771816</td>
</tr>
<tr>
<td>80 FZ</td>
<td>-0.01598371</td>
</tr>
<tr>
<td>38 FZ</td>
<td>-0.01598371</td>
</tr>
<tr>
<td>81 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>39 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>82 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>40 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>83 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>41 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>84 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>42 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>85 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>43 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>86 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>44 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>87 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>45 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>88 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>46 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>89 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>47 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>90 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>48 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>91 FZ</td>
<td>-0.1937751</td>
</tr>
<tr>
<td>49 FZ</td>
<td>-0.1937751</td>
</tr>
<tr>
<td>92 FZ</td>
<td>-0.5238495</td>
</tr>
<tr>
<td>50 FZ</td>
<td>-0.5238495</td>
</tr>
<tr>
<td>93 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>51 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>94 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>52 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>95 FZ</td>
<td>-0.2627459</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>53 FZ</th>
<th>-0.2627459</th>
</tr>
</thead>
<tbody>
<tr>
<td>96 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>54 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>99 FZ</td>
<td>-0.2849151</td>
</tr>
<tr>
<td>57 FZ</td>
<td>-0.2849151</td>
</tr>
<tr>
<td>102 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>60 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>103 FZ</td>
<td>-0.2446821</td>
</tr>
<tr>
<td>61 FZ</td>
<td>-0.2446821</td>
</tr>
<tr>
<td>104 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>62 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>105 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>63 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>106 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>64 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>107 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>65 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>108 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>66 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>109 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>67 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>110 FZ</td>
<td>-0.3982242</td>
</tr>
<tr>
<td>68 FZ</td>
<td>-0.3982242</td>
</tr>
<tr>
<td>111 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>69 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>112 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>70 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>113 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>71 FZ</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>114 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>72 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>115 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>73 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>116 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>74 FZ</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>117 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>75 FZ</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>118 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>76 FZ</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>119 FZ</td>
<td>-0.5197442</td>
</tr>
<tr>
<td>77 FZ</td>
<td>-0.5197442</td>
</tr>
<tr>
<td>120 FX</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>78 FX</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>121 FX</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>79 FX</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>122 FX</td>
<td>-0.2397556</td>
</tr>
<tr>
<td>80 FX</td>
<td>-0.2397556</td>
</tr>
<tr>
<td>123 FX</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>81 FX</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>124 FX</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>82 FX</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>125 FX</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>83 FX</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>126 FX</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>84 FX</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>127 FX</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>85 FX</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>128 FX</td>
<td>-0.1313729</td>
</tr>
</tbody>
</table>
Application Examples
EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>FZ</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>86</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>129</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>87</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>130</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>88</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>131</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>89</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>132</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>90</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>133</td>
<td>-0.1937751</td>
</tr>
<tr>
<td>91</td>
<td>-0.1937751</td>
</tr>
<tr>
<td>134</td>
<td>-0.5238495</td>
</tr>
<tr>
<td>92</td>
<td>-0.5238495</td>
</tr>
<tr>
<td>135</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>93</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>136</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>94</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>137</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>95</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>138</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>96</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>141</td>
<td>-0.2849151</td>
</tr>
<tr>
<td>99</td>
<td>-0.2849151</td>
</tr>
<tr>
<td>144</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>102</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>145</td>
<td>-0.2446821</td>
</tr>
<tr>
<td>103</td>
<td>-0.2446821</td>
</tr>
<tr>
<td>146</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>104</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>147</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>105</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>148</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>106</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>149</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>107</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>150</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>108</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>151</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>109</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>152</td>
<td>-0.3982242</td>
</tr>
<tr>
<td>110</td>
<td>-0.3982242</td>
</tr>
<tr>
<td>153</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>111</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>154</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>112</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>155</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>113</td>
<td>-0.1313729</td>
</tr>
<tr>
<td>156</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>114</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>157</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>115</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>158</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>116</td>
<td>-0.2627459</td>
</tr>
<tr>
<td>159</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>117</td>
<td>-0.1157724</td>
</tr>
<tr>
<td>160</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>118</td>
<td>-6.075998E-02</td>
</tr>
<tr>
<td>161</td>
<td>-0.5197442</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>119</td>
<td>FZ</td>
</tr>
<tr>
<td>162</td>
<td>FX</td>
</tr>
<tr>
<td>162</td>
<td>FX</td>
</tr>
<tr>
<td>163</td>
<td>FX</td>
</tr>
<tr>
<td>163</td>
<td>FX</td>
</tr>
<tr>
<td>164</td>
<td>FX</td>
</tr>
<tr>
<td>164</td>
<td>FX</td>
</tr>
<tr>
<td>165</td>
<td>FZ</td>
</tr>
<tr>
<td>123</td>
<td>FZ</td>
</tr>
<tr>
<td>166</td>
<td>FZ</td>
</tr>
<tr>
<td>124</td>
<td>FZ</td>
</tr>
<tr>
<td>167</td>
<td>FZ</td>
</tr>
<tr>
<td>125</td>
<td>FZ</td>
</tr>
<tr>
<td>168</td>
<td>FZ</td>
</tr>
<tr>
<td>126</td>
<td>FZ</td>
</tr>
<tr>
<td>169</td>
<td>FZ</td>
</tr>
<tr>
<td>127</td>
<td>FZ</td>
</tr>
<tr>
<td>170</td>
<td>FZ</td>
</tr>
<tr>
<td>128</td>
<td>FZ</td>
</tr>
<tr>
<td>171</td>
<td>FZ</td>
</tr>
<tr>
<td>129</td>
<td>FZ</td>
</tr>
<tr>
<td>172</td>
<td>FZ</td>
</tr>
<tr>
<td>130</td>
<td>FZ</td>
</tr>
<tr>
<td>173</td>
<td>FZ</td>
</tr>
<tr>
<td>131</td>
<td>FZ</td>
</tr>
<tr>
<td>174</td>
<td>FZ</td>
</tr>
<tr>
<td>132</td>
<td>FZ</td>
</tr>
<tr>
<td>175</td>
<td>FZ</td>
</tr>
<tr>
<td>133</td>
<td>FZ</td>
</tr>
<tr>
<td>176</td>
<td>FZ</td>
</tr>
<tr>
<td>135</td>
<td>FZ</td>
</tr>
<tr>
<td>177</td>
<td>FZ</td>
</tr>
<tr>
<td>136</td>
<td>FZ</td>
</tr>
<tr>
<td>178</td>
<td>FZ</td>
</tr>
<tr>
<td>137</td>
<td>FZ</td>
</tr>
<tr>
<td>179</td>
<td>FZ</td>
</tr>
<tr>
<td>138</td>
<td>FZ</td>
</tr>
<tr>
<td>182</td>
<td>FZ</td>
</tr>
<tr>
<td>141</td>
<td>FZ</td>
</tr>
<tr>
<td>185</td>
<td>FZ</td>
</tr>
<tr>
<td>144</td>
<td>FZ</td>
</tr>
<tr>
<td>186</td>
<td>FZ</td>
</tr>
<tr>
<td>146</td>
<td>FZ</td>
</tr>
<tr>
<td>187</td>
<td>FZ</td>
</tr>
<tr>
<td>147</td>
<td>FZ</td>
</tr>
<tr>
<td>188</td>
<td>FZ</td>
</tr>
<tr>
<td>148</td>
<td>FZ</td>
</tr>
<tr>
<td>189</td>
<td>FZ</td>
</tr>
<tr>
<td>149</td>
<td>FZ</td>
</tr>
<tr>
<td>190</td>
<td>FZ</td>
</tr>
<tr>
<td>150</td>
<td>FZ</td>
</tr>
<tr>
<td>191</td>
<td>FZ</td>
</tr>
<tr>
<td>151</td>
<td>FZ</td>
</tr>
<tr>
<td>192</td>
<td>FZ</td>
</tr>
<tr>
<td>152</td>
<td>FZ</td>
</tr>
<tr>
<td>193</td>
<td>FZ</td>
</tr>
<tr>
<td>153</td>
<td>FZ</td>
</tr>
<tr>
<td>194</td>
<td>FZ</td>
</tr>
</tbody>
</table>
### Application Examples

EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Node</th>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>154</td>
<td>FZ</td>
<td>-6.171203E-02</td>
</tr>
<tr>
<td>195</td>
<td>FZ</td>
<td>-0.1416653</td>
</tr>
<tr>
<td>155</td>
<td>FZ</td>
<td>-0.1334314</td>
</tr>
<tr>
<td>196</td>
<td>FZ</td>
<td>-0.2833306</td>
</tr>
<tr>
<td>156</td>
<td>FZ</td>
<td>-0.2668628</td>
</tr>
<tr>
<td>197</td>
<td>FZ</td>
<td>-0.2833306</td>
</tr>
<tr>
<td>157</td>
<td>FZ</td>
<td>-0.2668628</td>
</tr>
<tr>
<td>198</td>
<td>FZ</td>
<td>-0.2833306</td>
</tr>
<tr>
<td>158</td>
<td>FZ</td>
<td>-0.2668628</td>
</tr>
<tr>
<td>199</td>
<td>FZ</td>
<td>-0.1248425</td>
</tr>
<tr>
<td>159</td>
<td>FZ</td>
<td>-0.1175864</td>
</tr>
<tr>
<td>200</td>
<td>FZ</td>
<td>-6.552019E-02</td>
</tr>
<tr>
<td>160</td>
<td>FZ</td>
<td>-6.171203E-02</td>
</tr>
<tr>
<td>201</td>
<td>FZ</td>
<td>-0.2594245</td>
</tr>
<tr>
<td>161</td>
<td>FZ</td>
<td>-0.2443463</td>
</tr>
<tr>
<td>202</td>
<td>FX</td>
<td>-0.1248425</td>
</tr>
<tr>
<td>203</td>
<td>FX</td>
<td>-0.2585391</td>
</tr>
<tr>
<td>204</td>
<td>FX</td>
<td>-0.1248425</td>
</tr>
</tbody>
</table>

**LOAD 9 LOADTYPE Dead TITLE DEAD LOAD**

SELFWEIGHT Y -1

**JOINT LOAD**

<table>
<thead>
<tr>
<th>Joint</th>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>92</td>
<td>FY</td>
<td>-0.15622E-14</td>
</tr>
<tr>
<td>183</td>
<td>FY</td>
<td>-1.06989E-13</td>
</tr>
<tr>
<td>110</td>
<td>FY</td>
<td>-5.36661E-15</td>
</tr>
<tr>
<td>119</td>
<td>FY</td>
<td>-2.300696E-14</td>
</tr>
<tr>
<td>134</td>
<td>FY</td>
<td>-8.15622E-14</td>
</tr>
<tr>
<td>145</td>
<td>FY</td>
<td>-1.06989E-13</td>
</tr>
<tr>
<td>152</td>
<td>FY</td>
<td>-5.36661E-15</td>
</tr>
<tr>
<td>161</td>
<td>FY</td>
<td>-2.300696E-14</td>
</tr>
<tr>
<td>192</td>
<td>FY</td>
<td>-5.36661E-15</td>
</tr>
<tr>
<td>201</td>
<td>FY</td>
<td>-2.300696E-14</td>
</tr>
</tbody>
</table>

**MEMBER LOAD**

<table>
<thead>
<tr>
<th>Member</th>
<th>Force Component</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>151</td>
<td>UNI Y</td>
<td>-0.77875</td>
</tr>
<tr>
<td>152</td>
<td>UNI Y</td>
<td>-0.77875</td>
</tr>
<tr>
<td>153</td>
<td>UNI Y</td>
<td>-0.77875</td>
</tr>
<tr>
<td>154</td>
<td>UNI Y</td>
<td>-0.77875</td>
</tr>
<tr>
<td>155</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>156</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>157</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>158</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>159</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>160</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>161</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>162</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>163</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>164</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>166</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>167</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>168</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>169</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>170</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>171</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>172</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>173</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
<tr>
<td>174</td>
<td>UNI Y</td>
<td>-0.39175</td>
</tr>
</tbody>
</table>
Application Examples

EX. Interactive Concrete Design Examples

175 UNI Y -0.77875
176 UNI Y -0.77875
177 UNI Y -0.77875
178 UNI Y -0.77875
179 UNI Y -0.77875
180 UNI Y -0.77875
181 UNI Y -0.77875
182 UNI Y -0.77875
183 UNI Y -0.77875
184 UNI Y -0.77875
185 UNI Y -0.39175
186 UNI Y -0.39175
187 UNI Y -0.39175
188 UNI Y -0.39175
189 UNI Y -0.39175
190 UNI Y -0.39175
191 UNI Y -0.39175
192 UNI Y -0.39175
193 UNI Y -0.39175
194 UNI Y -0.39175
195 UNI Y -0.39175
196 UNI Y -0.39175
197 UNI Y -0.39175
198 UNI Y -0.39175
199 UNI Y -0.39175
201 UNI Y -0.39175
202 UNI Y -0.39175
204 UNI Y -0.39175
205 UNI Y -0.39175
206 UNI Y -0.77875
207 UNI Y -0.77875
208 UNI Y -0.39175
209 UNI Y -0.77875
210 UNI Y -0.77875
211 UNI Y -0.39175
212 UNI Y -0.77875
213 UNI Y -0.77875
217 UNI Y -0.77875
217 TRAP Y -1.642 0 5.145 7.998
217 UNI Y -1.642 2.855 5.143
217 TRAP Y 0 -1.642 0 2.854
218 UNI Y -0.77875
218 TRAP Y -1.642 0 5.145 7.998
218 UNI Y -1.642 2.855 5.143
218 TRAP Y 0 -1.642 0 2.854
219 UNI Y -0.77875
219 TRAP Y -1.642 0 5.145 7.998
219 UNI Y -1.642 2.855 5.143
219 TRAP Y 0 -1.642 0 2.854
220 UNI Y -0.77875
220 TRAP Y -1.642 0 5.145 7.998
220 UNI Y -1.642 2.855 5.143
220 TRAP Y 0 -1.642 0 2.854
221 UNI Y -0.39175
221 TRAP Y -1.642 0 5.145 7.998
221 UNI Y -1.642 2.855 5.143
221 TRAP Y 0 -1.642 0 2.854
221 TRAP Y 0 -2.8 0 3.998
Application Examples
EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Step</th>
<th>Action</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>RotX</th>
<th>RotY</th>
<th>RotZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>233</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.302</td>
<td>1.745</td>
<td>0</td>
<td>0.801</td>
<td></td>
</tr>
<tr>
<td>234</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>234</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>234</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>234</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>234</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>235</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>236</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>236</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>236</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>236</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>237</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>238</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>238</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.59</td>
<td>0</td>
<td>3.574</td>
<td></td>
</tr>
<tr>
<td>238</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.59</td>
<td>0</td>
<td>-0.731</td>
<td>3.576</td>
<td>5.828</td>
</tr>
<tr>
<td>239</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>239</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.816</td>
<td>0</td>
<td>0.988</td>
<td>1.918</td>
<td></td>
</tr>
<tr>
<td>239</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.816</td>
<td>0</td>
<td>0.986</td>
<td></td>
</tr>
<tr>
<td>239</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.731</td>
<td>0</td>
<td>0</td>
<td>1.918</td>
<td></td>
</tr>
<tr>
<td>240</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>240</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.676</td>
<td>0</td>
<td>2.913</td>
<td></td>
</tr>
<tr>
<td>240</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.816</td>
<td>0</td>
<td>2.915</td>
<td>5.828</td>
</tr>
<tr>
<td>240</td>
<td>UNI</td>
<td>Y</td>
<td>-0.56</td>
<td>0</td>
<td>5.828</td>
<td></td>
<td></td>
</tr>
<tr>
<td>241</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>241</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>241</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
</tr>
<tr>
<td>242</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>242</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>242</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
</tr>
<tr>
<td>243</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>243</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>243</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
</tr>
<tr>
<td>244</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>244</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>244</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>245</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>245</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.676</td>
<td>0</td>
<td>2.913</td>
<td></td>
</tr>
<tr>
<td>245</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.816</td>
<td>0</td>
<td>2.915</td>
<td>5.828</td>
</tr>
<tr>
<td>246</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>246</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.106</td>
<td>0</td>
<td>2.247</td>
<td></td>
</tr>
<tr>
<td>246</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.106</td>
<td>0</td>
<td>2.248</td>
<td>4.848</td>
</tr>
<tr>
<td>247</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>247</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>247</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>9.948</td>
</tr>
<tr>
<td>248</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>248</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>248</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>4</td>
<td>5.048</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>Step</th>
<th>Action</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>248</td>
<td>TRAP</td>
<td>Y -2.8 0 5.05 9.049</td>
</tr>
<tr>
<td>249</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>249</td>
<td>UNI</td>
<td>Y -2.8 4 4.898</td>
</tr>
<tr>
<td>249</td>
<td>TRAP</td>
<td>Y -2.8 0 4.9 8.899</td>
</tr>
<tr>
<td>250</td>
<td>UNI</td>
<td>Y -2.8 4 4.898</td>
</tr>
<tr>
<td>250</td>
<td>TRAP</td>
<td>Y -1.642 0 2.855 5.709</td>
</tr>
<tr>
<td>250</td>
<td>TRAP</td>
<td>Y 0 -1.642 0 2.854</td>
</tr>
<tr>
<td>251</td>
<td>UNI</td>
<td>Y -0.77875</td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>Y -2.8 0 3.998</td>
</tr>
<tr>
<td>251</td>
<td>UNI</td>
<td>Y -2.8 4 4.948</td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>Y -2.8 0 4.95 8.948</td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>Y -2.8 4 4.948</td>
</tr>
<tr>
<td>252</td>
<td>UNI</td>
<td>Y -0.39175</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>Y -2.8 4 5.048</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>Y -2.8 0 5.05 9.049</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>Y -2.8 4 5.048</td>
</tr>
<tr>
<td>253</td>
<td>UNI</td>
<td>Y -2.8 4 4.898</td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>Y -2.8 0 4.9 8.899</td>
</tr>
<tr>
<td>253</td>
<td>UNI</td>
<td>Y -2.8 4 5.048</td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>Y -2.8 4 5.048</td>
</tr>
<tr>
<td>254</td>
<td>UNI</td>
<td>Y -2.8 4 4.898</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>Y -2.8 0 4.9 8.899</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>254</td>
<td>UNI</td>
<td>Y -1.642 0 2.855 5.709</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>Y 0 -1.642 0 2.854</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>Y -1.642 0 2.855 5.709</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>Y 0 -1.642 0 2.854</td>
</tr>
<tr>
<td>255</td>
<td>UNI</td>
<td>Y -0.39175</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>Y -2.8 4 4.948</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>Y -2.8 0 4.95 8.948</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>Y -2.8 4 4.948</td>
</tr>
<tr>
<td>256</td>
<td>UNI</td>
<td>Y -2.8 4 5.048</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>Y -2.8 0 5.05 9.049</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>Y -2.8 4 5.048</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>Y -2.8 0 5.05 9.049</td>
</tr>
<tr>
<td>257</td>
<td>UNI</td>
<td>Y -0.39175</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>Y -2.8 4 4.898</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>Y -2.8 0 4.9 8.899</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>Y 0 -2.8 0 3.998</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>Y -2.8 4 4.898</td>
</tr>
<tr>
<td>258</td>
<td>UNI</td>
<td>Y -0.39175</td>
</tr>
<tr>
<td>258</td>
<td>TRAP</td>
<td>Y -1.642 0 2.855 5.709</td>
</tr>
<tr>
<td>258</td>
<td>TRAP</td>
<td>Y 0 -1.642 0 2.854</td>
</tr>
</tbody>
</table>

*STAAD.Pro 5008 User Manual*
Application Examples
EX. Interactive Concrete Design Examples

258 TRAP  Y  -1.642 0 2.855 5.709
258 TRAP  Y  0 -1.642 0 2.854
259 UNI  Y  -0.39175
259 TRAP  Y  0 -2.8 0 3.998
259 UNI  Y  -2.8 4 4.948
259 TRAP  Y  -2.8 0 4.95 8.948
260 UNI  Y  -0.39175
260 TRAP  Y  0 -2.8 0 3.998
260 UNI  Y  -2.8 4 5.048
260 TRAP  Y  -2.8 0 5.05 9.049
260 UNI  Y  -2.8 4 5.048
260 TRAP  Y  -2.8 0 5.05 9.049
260 UNI  Y  -0.39175
261 TRAP  Y  0 -2.8 0 3.998
261 UNI  Y  -2.8 4 4.898
261 TRAP  Y  -2.8 -1.743 4.95 6.458
261 TRAP  Y  0 -1.676 0 2.913
261 TRAP  Y  -1.676 2.915 3.543
261 TRAP  Y  -1.676 0 3.545 6.459
264 UNI  Y  -0.39175
264 TRAP  Y  -1.743 0 0 2.489
265 UNI  Y  -0.39175
265 TRAP  Y  0 -2.8 0 3.998
265 UNI  Y  -2.8 4 5.048
266 TRAP  Y  -2.8 -2.275 5.95 5.799
265 TRAP  Y  -1.968 0 2.844 5.798
266 TRAP  Y  0 -1.968 0 2.842
266 UNI  Y  -0.472 1.18 1.624
266 TRAP  Y  -0.472 0 1.178
267 UNI  Y  -0.39175
267 TRAP  Y  0 -2.8 0 3.998
267 UNI  Y  -2.8 4 4.898
267 TRAP  Y  -2.8 0 4.9 8.899
267 TRAP  Y  -0.974 0 3.89 8.899
267 TRAP  Y  0 -0.974 0 3.889
268 UNI  Y  -0.39175
268 TRAP  Y  -1.642 0 2.855 5.709
268 TRAP  Y  0 -1.642 0 2.854
268 TRAP  Y  -0.249 0 1.952 5.709
268 TRAP  Y  0 -0.249 0 1.95
269 UNI  Y  -0.472 1.18 1.624
Application Examples
EX. Interactive Concrete Design Examples

269 TRAP Y 0 -0.472 0 1.178
269 TRAP Y -0.598 -0.58 1.572 1.624
269 TRAP Y 0 -0.598 0 1.571
270 UNI Y -0.39175
270 TRAP Y 0 -1.676 0 2.913
270 UNI Y -1.676 2.915 3.543
270 TRAP Y -1.676 0 3.545 6.459
270 TRAP Y -0.989 -0.49 3.921 6.459
270 TRAP Y 0 -0.989 0 3.919
271 UNI Y -0.39175
271 TRAP Y -0.49 0 0 2.489
272 UNI Y -0.77875
272 TRAP Y 0 -3.711 0 4.841
272 TRAP Y -3.711 0 4.842 10.013
273 UNI Y -0.77875
273 TRAP Y -3.672 0 5.373 10.253
273 TRAP Y 0 -3.672 0 5.371
274 UNI Y -0.39175
274 TRAP Y -0.569 0 1.498 3.417
274 TRAP Y 0 -0.569 0 1.496
274 TRAP Y 0 -3.085 0 3.417
275 UNI Y -0.77875
275 TRAP Y -0.237 0 3.779 6.007
275 TRAP Y 0 -0.237 0 3.778
276 UNI Y -0.77875
276 TRAP Y -0.94 0 4.815 9.416
276 TRAP Y 0 -0.94 0 4.813
277 UNI Y -0.39175
277 TRAP Y -3.999 0 1.131 6.102
277 TRAP Y -3.085 -3.999 0 1.13
277 TRAP Y -1.87 0 2.629 6.102
277 TRAP Y 0 -1.87 0 2.628
278 UNI Y -0.77875
278 TRAP Y -0.926 0 4.8 9.365
278 TRAP Y 0 -0.926 0 4.799
279 UNI Y -0.77875
279 TRAP Y 0 -4.08 0 4.661
279 TRAP Y -4.08 0 4.663 8.996
280 TRAP Y -1.138 0 0 1.624
280 TRAP Y -0.472 0 0.445 1.623
280 UNI Y -0.472 0 0.443
281 TRAP Y -1.111 -2.09 0 1.038
281 TRAP Y -2.09 -1.976 1.04 1.178
281 TRAP Y -0.472 0 0 1.178
282 TRAP Y -0.472 0 0.445 1.623
282 UNI Y -0.472 0 0.443
282 TRAP Y -0.58 0 0 1.624
283 UNI Y -0.77875
283 TRAP Y -1.642 0 5.145 7.998
283 UNI Y -1.642 2.855 5.143
283 TRAP Y 0 -1.642 0 2.854
284 UNI Y -0.77875
284 TRAP Y -1.642 0 5.145 7.998
284 UNI Y -1.642 2.855 5.143
284 TRAP Y 0 -1.642 0 2.854
285 UNI Y -0.77875
285 TRAP Y -1.642 0 5.145 7.998
285 UNI Y -1.642 2.855 5.143
Application Examples
EX. Interactive Concrete Design Examples

| 285 TRAP Y 0 -1.642 0 2.854 |
| 286 UNI Y -0.77875 |
| 286 TRAP Y -1.642 0 5.145 7.998 |
| 286 UNI Y -1.642 2.855 5.143 |
| 286 TRAP Y 0 -1.642 0 2.854 |
| 287 UNI Y -0.39175 |
| 287 TRAP Y -1.642 0 5.145 7.998 |
| 287 UNI Y -1.642 2.855 5.143 |
| 287 TRAP Y 0 -1.642 0 2.854 |
| 287 TRAP Y -2.8 0 3.998 |
| 287 TRAP Y -2.8 0 4 7.998 |
| 288 UNI Y -0.39175 |
| 288 TRAP Y -1.642 0 5.145 7.998 |
| 288 UNI Y -1.642 2.855 5.143 |
| 288 TRAP Y 0 -1.642 0 2.854 |
| 288 TRAP Y -2.8 0 3.998 |
| 288 TRAP Y -2.8 0 4 7.998 |
| 289 UNI Y -0.39175 |
| 289 TRAP Y -1.642 0 5.145 7.998 |
| 289 UNI Y -1.642 2.855 5.143 |
| 289 TRAP Y 0 -1.642 0 2.854 |
| 289 TRAP Y -2.8 0 3.998 |
| 289 TRAP Y -2.8 0 4 7.998 |
| 290 UNI Y -0.39175 |
| 290 TRAP Y -1.642 0 5.145 7.998 |
| 290 UNI Y -1.642 2.855 5.143 |
| 290 TRAP Y 0 -1.642 0 2.854 |
| 290 TRAP Y -2.8 0 3.998 |
| 290 TRAP Y -2.8 0 4 7.998 |
| 291 UNI Y -0.39175 |
| 291 TRAP Y 0 -0.761 0 0.874 |
| 291 TRAP Y -0.761 0 0.876 1.869 |
| 291 TRAP Y -2.805 0 0.961 1.869 |
| 291 TRAP Y 0 -2.805 0 0.96 |
| 292 UNI Y -0.39175 |
| 292 TRAP Y 0 -2.8 0 3.998 |
| 292 TRAP Y -2.8 0 5 7.998 |
| 292 TRAP Y 0 -2.8 0 3.998 |
| 292 TRAP Y -2.8 0 4 7.998 |
| 293 UNI Y -0.39175 |
| 293 TRAP Y 0 -2.8 0 3.998 |
| 293 TRAP Y -2.8 0 4 7.998 |
| 293 TRAP Y 0 -2.8 0 3.998 |
| 293 TRAP Y -2.8 0 4 7.998 |
| 294 UNI Y -0.39175 |
| 294 TRAP Y 0 -2.8 0 3.998 |
| 294 TRAP Y -2.8 0 4 7.998 |
| 294 TRAP Y 0 -2.8 0 3.998 |
| 294 TRAP Y -2.8 0 4 7.998 |
| 295 UNI Y -0.39175 |
| 295 TRAP Y 0 -2.8 0 3.998 |
| 295 TRAP Y -2.8 0 4 7.998 |
| 295 TRAP Y 0 -2.8 0 3.998 |
| 295 TRAP Y -2.8 0 4 7.998 |
| 296 TRAP Y 0 -1.111 0 1.179 |
| 296 TRAP Y 0 -0.472 0 1.178 |
| 297 UNI Y -0.39175 |
| 297 TRAP Y -1.976 0 0 2.429 |
### Application Examples

EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Type</th>
<th>Y</th>
<th>X1</th>
<th>X2</th>
<th>X3</th>
<th>X4</th>
</tr>
</thead>
<tbody>
<tr>
<td>297</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.689</td>
<td>1.307</td>
<td>2.428</td>
<td></td>
</tr>
<tr>
<td>297</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.689</td>
<td>1.306</td>
<td></td>
</tr>
<tr>
<td>298</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.39175</td>
<td></td>
<td></td>
</tr>
<tr>
<td>298</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.472</td>
<td>1.178</td>
<td></td>
</tr>
<tr>
<td>298</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.302</td>
<td>2.358</td>
<td></td>
</tr>
<tr>
<td>299</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.39175</td>
<td></td>
<td></td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.611</td>
<td>1.603</td>
<td></td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.611</td>
<td>1.605</td>
<td>3.489</td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.745</td>
<td>0</td>
<td>3.488</td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.302</td>
<td>-1.745</td>
<td>0</td>
<td>0.801</td>
</tr>
<tr>
<td>300</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.39175</td>
<td></td>
<td></td>
</tr>
<tr>
<td>300</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0.998</td>
<td></td>
</tr>
<tr>
<td>300</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>300</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>301</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.39175</td>
<td></td>
<td></td>
</tr>
<tr>
<td>301</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>301</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>302</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.39175</td>
<td></td>
<td></td>
</tr>
<tr>
<td>302</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>302</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>303</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.39175</td>
<td></td>
<td></td>
</tr>
<tr>
<td>303</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>303</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>304</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.39175</td>
<td></td>
<td></td>
</tr>
<tr>
<td>304</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.676</td>
<td>0</td>
<td>2.913</td>
<td></td>
</tr>
<tr>
<td>305</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-1.676</td>
<td>0</td>
<td>2.915</td>
</tr>
<tr>
<td>305</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.676</td>
<td>0</td>
<td>2.915</td>
<td>5.828</td>
</tr>
<tr>
<td>306</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.56</td>
<td>5</td>
<td>8.28</td>
</tr>
<tr>
<td>306</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>306</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>307</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.77875</td>
<td></td>
<td></td>
</tr>
<tr>
<td>307</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>308</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>308</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>309</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.77875</td>
<td></td>
<td></td>
</tr>
<tr>
<td>309</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>309</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>310</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.77875</td>
<td></td>
<td></td>
</tr>
<tr>
<td>310</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>310</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>7.998</td>
<td></td>
</tr>
<tr>
<td>311</td>
<td>UNI</td>
<td>Y</td>
<td>0</td>
<td>-0.77875</td>
<td></td>
<td></td>
</tr>
<tr>
<td>311</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.676</td>
<td>0</td>
<td>2.913</td>
<td></td>
</tr>
<tr>
<td>311</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.676</td>
<td>0</td>
<td>2.915</td>
<td>5.828</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>312</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>312</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.106</td>
<td>0</td>
<td>2.247</td>
</tr>
<tr>
<td>312</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.106</td>
<td>0</td>
<td>2.248</td>
<td>8.484</td>
</tr>
<tr>
<td>313</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>313</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
</tr>
<tr>
<td>313</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>4.948</td>
<td></td>
</tr>
<tr>
<td>313</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4.95</td>
<td>8.948</td>
</tr>
<tr>
<td>314</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>5.048</td>
<td></td>
</tr>
<tr>
<td>314</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>5.05</td>
<td>9.049</td>
</tr>
<tr>
<td>315</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>315</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
</tr>
<tr>
<td>315</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>4.898</td>
<td></td>
</tr>
<tr>
<td>315</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4.9</td>
<td>8.899</td>
</tr>
<tr>
<td>316</td>
<td>UNI</td>
<td>Y</td>
<td>-0.77875</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>316</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642</td>
<td>0</td>
<td>2.855</td>
<td>5.709</td>
</tr>
<tr>
<td>316</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.642</td>
<td>0</td>
<td>2.854</td>
</tr>
<tr>
<td>317</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>317</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
</tr>
<tr>
<td>317</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>4.948</td>
<td></td>
</tr>
<tr>
<td>317</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4.95</td>
<td>8.948</td>
</tr>
<tr>
<td>318</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>5.048</td>
<td></td>
</tr>
<tr>
<td>318</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>5.05</td>
<td>9.049</td>
</tr>
<tr>
<td>319</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>319</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
</tr>
<tr>
<td>319</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>4.898</td>
<td></td>
</tr>
<tr>
<td>319</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4.9</td>
<td>8.899</td>
</tr>
<tr>
<td>319</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>319</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>320</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>320</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642</td>
<td>0</td>
<td>2.855</td>
<td>5.709</td>
</tr>
<tr>
<td>320</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.642</td>
<td>0</td>
<td>2.854</td>
</tr>
<tr>
<td>321</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>321</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
</tr>
<tr>
<td>321</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>4.948</td>
<td></td>
</tr>
<tr>
<td>321</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>4.95</td>
<td>8.948</td>
</tr>
<tr>
<td>322</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
<td>5.048</td>
<td></td>
</tr>
<tr>
<td>322</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>5.05</td>
<td>9.049</td>
</tr>
<tr>
<td>322</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
<td>5.05</td>
<td>9.049</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>323</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>323</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 4.898</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 4.9 8.899</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>323</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 4.898</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 4.9 8.899</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>324</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642 0 2.855 5.709</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642 0 2.854</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642 0 2.855 5.709</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642 0 2.854</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>325</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>325</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>325</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 4.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>325</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 4.95 8.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>325</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>325</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 4.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>325</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 4.95 8.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>326</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>326</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>326</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 5.048</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>326</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 5.05 9.049</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>326</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>326</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 5.048</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>326</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 5.05 9.049</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>327</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>327</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>327</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 4.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>327</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 4.9 8.899</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>327</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>327</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 4.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>327</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 0 4.9 8.899</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>328</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>328</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642 0 2.855 5.709</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>328</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642 0 2.854</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>328</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642 0 2.855 5.709</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>328</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642 0 2.854</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>329</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>329</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>329</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 4.948</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>329</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 -1.743 4.95 6.458</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>329</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.676 0 2.913</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>329</td>
<td>UNI</td>
<td>Y</td>
<td>-1.676 2.915 3.543</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>329</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.676 0 3.545 6.459</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>330</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>330</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.743 0 0 2.489</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>331</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>331</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>331</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8 4 5.048</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>331</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8 -2.275 5.05 5.799</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>331</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.968 0 2.844 5.798</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>331</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.968 0 2.842</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>332</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.275 -1.138 0 1.624</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>332</td>
<td>UNI</td>
<td>Y</td>
<td>-0.472 1.18 1.624</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>332</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.472 0 1.178</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>333</td>
<td>UNI</td>
<td>Y</td>
<td>-0.39175</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>333</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8 0 3.998</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>UNI</th>
<th>TRAP</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| 333 | UNI  | Y  -2.8  4.898 |
| 333 | TRAP | Y  -2.8  0.9  8.899 |
| 333 | TRAP | Y  -0.974  0.39  8.899 |
| 333 | TRAP | Y  0  -0.974  0.389 |
| 334 | UNI  | Y  -3.9175 |
| 334 | TRAP | Y  -1.642  0.2855  5.709 |
| 334 | TRAP | Y  0  -1.642  0.2854 |
| 334 | TRAP | Y  -0.124  0.95  5.709 |
| 334 | TRAP | Y  0  -0.124  0.95 |
| 335 | UNI  | Y  -0.472  1.18  1.624 |
| 335 | TRAP | Y  -0.472  0.1178 |
| 335 | TRAP | Y  -0.598  -0.58  1.572  1.624 |
| 335 | TRAP | Y  0  -0.598  0.1571 |
| 336 | UNI  | Y  -0.39175 |
| 336 | TRAP | Y  -1.676  2.915  3.543 |
| 336 | TRAP | Y  -1.676  0.3545  6.459 |
| 336 | TRAP | Y  -0.989  -0.49  3.921  6.459 |
| 336 | TRAP | Y  0  -0.989  0.3919 |
| 337 | UNI  | Y  -0.39175 |
| 337 | TRAP | Y  -0.49  0  2.489 |
| 338 | UNI  | Y  -0.77875 |
| 338 | TRAP | Y  -3.711  0.4841 |
| 338 | TRAP | Y  -3.711  0.4842  10.013 |
| 339 | UNI  | Y  -0.77875 |
| 339 | TRAP | Y  -3.672  0.573  10.253 |
| 339 | TRAP | Y  0  -3.672  0.571 |
| 340 | UNI  | Y  -0.39175 |
| 340 | TRAP | Y  -0.569  0.1498  3.417 |
| 340 | TRAP | Y  0  -0.569  0.1496 |
| 340 | TRAP | Y  0  -3.005  0.3417 |
| 341 | UNI  | Y  -0.77875 |
| 341 | TRAP | Y  -0.237  0.3779  6.007 |
| 341 | TRAP | Y  0  -0.237  0.3778 |
| 342 | UNI  | Y  -0.77875 |
| 342 | TRAP | Y  -0.94  0.4815  9.416 |
| 342 | TRAP | Y  0  -0.94  0.4813 |
| 343 | UNI  | Y  -0.39175 |
| 343 | TRAP | Y  -3.999  0.131  6.102 |
| 343 | TRAP | Y  -3.005  -3.999  0.113 |
| 343 | TRAP | Y  -1.87  0.2629  6.102 |
| 343 | TRAP | Y  0  -1.87  0.2628 |
| 344 | UNI  | Y  -0.77875 |
| 344 | TRAP | Y  -0.926  0.48  9.365 |
| 344 | TRAP | Y  0  -0.926  0.4799 |
| 345 | UNI  | Y  -0.77875 |
| 345 | TRAP | Y  -4.08  0.4661 |
| 345 | TRAP | Y  -4.08  0.4663  8.996 |
| 346 | TRAP | Y  -1.138  0  1.624 |
| 346 | TRAP | Y  -0.472  0  0.445  1.623 |
| 346 | UNI  | Y  -0.472  0  0.443 |
| 347 | TRAP | Y  -1.111  -2.09  0  1.038 |
| 347 | TRAP | Y  -2.09  -1.976  1.04  1.178 |
| 347 | TRAP | Y  -0.472  0  0  1.178 |
| 348 | TRAP | Y  -0.472  0  0  1.623 |
| 348 | TRAP | Y  -0.472  0  0.443 |
| 349 | TRAP | Y  -0.58  0  1.624 |
| 349 | TRAP | Y  -1.642  0.145  7.998 |
### Application Examples

#### EX. Interactive Concrete Design Examples

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>349</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>349</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>350</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642  0  5.145  7.998</td>
</tr>
<tr>
<td>350</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>350</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>351</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642  0  5.145  7.998</td>
</tr>
<tr>
<td>351</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>351</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>352</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642  0  5.145  7.998</td>
</tr>
<tr>
<td>352</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>352</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>353</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  5.145  7.998</td>
</tr>
<tr>
<td>353</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>353</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>353</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>354</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642  0  5.145  7.998</td>
</tr>
<tr>
<td>354</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>354</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>354</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>354</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>355</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642  0  5.145  7.998</td>
</tr>
<tr>
<td>355</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>355</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>355</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>355</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  4  7.998</td>
</tr>
<tr>
<td>356</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642  0  5.145  7.998</td>
</tr>
<tr>
<td>356</td>
<td>UNI</td>
<td>Y</td>
<td>-1.642  2.855  5.143</td>
</tr>
<tr>
<td>356</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.642  0  2.854</td>
</tr>
<tr>
<td>356</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.761  0  0.874</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.761  0  0.876  1.869</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.805  0  0.961  1.869</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.805  0  0.96</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>359</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>359</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>359</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>359</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -2.8  0  3.998</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8  0  4  7.998</td>
</tr>
<tr>
<td>362</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -1.111  0  1.179</td>
</tr>
<tr>
<td>362</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.472  0  1.178</td>
</tr>
<tr>
<td>363</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.976  0  0  2.429</td>
</tr>
<tr>
<td>363</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.689  0  1.307  2.428</td>
</tr>
<tr>
<td>363</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.689  0  1.396</td>
</tr>
<tr>
<td>364</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.472  0  1.178</td>
</tr>
<tr>
<td>364</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.472  0  1.18  2.358</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Type</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>W</th>
</tr>
</thead>
<tbody>
<tr>
<td>364</td>
<td>TRAP</td>
<td>-1.302</td>
<td>0</td>
<td>2.358</td>
<td></td>
</tr>
<tr>
<td>365</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>4.7998</td>
<td></td>
</tr>
<tr>
<td>366</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>4.7998</td>
<td></td>
</tr>
<tr>
<td>367</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>368</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>369</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>370</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>4.7998</td>
<td></td>
</tr>
<tr>
<td>371</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>4.7998</td>
<td></td>
</tr>
<tr>
<td>372</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>373</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>374</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>375</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>376</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>377</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>378</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>379</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>380</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>381</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>382</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
<tr>
<td>383</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>3.998</td>
<td></td>
</tr>
</tbody>
</table>

### STAAD.Pro 5017 User Manual
Application Examples
EX. Interactive Concrete Design Examples

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>383</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>384</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>384</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>384</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>384</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>384</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>385</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>385</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>385</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>385</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642</td>
<td>0</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.642</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642</td>
<td>0</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.642</td>
</tr>
<tr>
<td>387</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>387</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>387</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>387</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>387</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>387</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>388</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>388</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>388</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>388</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>388</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>388</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>389</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>389</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>389</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>389</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>389</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>389</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>390</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>390</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>390</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642</td>
<td>0</td>
</tr>
<tr>
<td>390</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.642</td>
</tr>
<tr>
<td>390</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642</td>
<td>0</td>
</tr>
<tr>
<td>390</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.642</td>
</tr>
<tr>
<td>391</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>391</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>391</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>391</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>391</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>391</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>391</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>392</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>392</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>393</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>393</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>393</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>393</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>393</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>393</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-2.8</td>
</tr>
<tr>
<td>393</td>
<td>UNI</td>
<td>Y</td>
<td>-2.8</td>
<td>4</td>
</tr>
<tr>
<td>393</td>
<td>TRAP</td>
<td>Y</td>
<td>-2.8</td>
<td>0</td>
</tr>
<tr>
<td>394</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.642</td>
<td>0</td>
</tr>
</tbody>
</table>
Application Examples

EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Step</th>
<th>Option</th>
<th>Y Component</th>
<th>X Component</th>
<th>Z Component</th>
<th>Load Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>394</td>
<td>TRAP</td>
<td>0</td>
<td>-1.642</td>
<td>2.854</td>
<td>10</td>
</tr>
<tr>
<td>394</td>
<td>TRAP</td>
<td>-1.642</td>
<td>2.855</td>
<td>5.709</td>
<td>10</td>
</tr>
<tr>
<td>394</td>
<td>TRAP</td>
<td>0</td>
<td>-1.642</td>
<td>2.854</td>
<td>10</td>
</tr>
<tr>
<td>395</td>
<td>TRAP</td>
<td>0</td>
<td>-2.8</td>
<td>3.998</td>
<td>10</td>
</tr>
<tr>
<td>395</td>
<td>UNI</td>
<td>-2.8</td>
<td>4</td>
<td>4.948</td>
<td>10</td>
</tr>
<tr>
<td>395</td>
<td>TRAP</td>
<td>-2.8</td>
<td>-1.743</td>
<td>4.95</td>
<td>10</td>
</tr>
<tr>
<td>395</td>
<td>TRAP</td>
<td>0</td>
<td>-1.676</td>
<td>2.913</td>
<td>10</td>
</tr>
<tr>
<td>395</td>
<td>UNI</td>
<td>-1.676</td>
<td>2.915</td>
<td>3.543</td>
<td>10</td>
</tr>
<tr>
<td>395</td>
<td>TRAP</td>
<td>-1.676</td>
<td>0</td>
<td>3.545</td>
<td>10</td>
</tr>
<tr>
<td>395</td>
<td>TRAP</td>
<td>-1.743</td>
<td>0</td>
<td>2.489</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>TRAP</td>
<td>0</td>
<td>-2.8</td>
<td>3.998</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>UNI</td>
<td>-2.8</td>
<td>4</td>
<td>5.048</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>TRAP</td>
<td>-2.8</td>
<td>-2.275</td>
<td>5.05</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>TRAP</td>
<td>-1.968</td>
<td>0</td>
<td>2.844</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>TRAP</td>
<td>0</td>
<td>-1.968</td>
<td>2.842</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>TRAP</td>
<td>-2.275</td>
<td>-1.138</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>UNI</td>
<td>-0.472</td>
<td>1.18</td>
<td>1.624</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>TRAP</td>
<td>0</td>
<td>-0.472</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>397</td>
<td>TRAP</td>
<td>0</td>
<td>-2.8</td>
<td>3.998</td>
<td>10</td>
</tr>
<tr>
<td>399</td>
<td>UNI</td>
<td>-2.8</td>
<td>4</td>
<td>5.048</td>
<td>10</td>
</tr>
<tr>
<td>399</td>
<td>TRAP</td>
<td>-2.8</td>
<td>0</td>
<td>4.9</td>
<td>10</td>
</tr>
<tr>
<td>399</td>
<td>TRAP</td>
<td>-0.974</td>
<td>0</td>
<td>3.89</td>
<td>10</td>
</tr>
<tr>
<td>399</td>
<td>TRAP</td>
<td>0</td>
<td>-0.974</td>
<td>3.889</td>
<td>10</td>
</tr>
<tr>
<td>400</td>
<td>TRAP</td>
<td>-1.642</td>
<td>0</td>
<td>2.855</td>
<td>10</td>
</tr>
<tr>
<td>400</td>
<td>TRAP</td>
<td>0</td>
<td>-1.642</td>
<td>2.854</td>
<td>10</td>
</tr>
<tr>
<td>400</td>
<td>TRAP</td>
<td>-0.249</td>
<td>0</td>
<td>1.952</td>
<td>10</td>
</tr>
<tr>
<td>400</td>
<td>TRAP</td>
<td>0</td>
<td>-0.249</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>401</td>
<td>UNI</td>
<td>-0.472</td>
<td>1.18</td>
<td>1.624</td>
<td>10</td>
</tr>
<tr>
<td>401</td>
<td>TRAP</td>
<td>0</td>
<td>-0.472</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>401</td>
<td>TRAP</td>
<td>-0.598</td>
<td>-0.58</td>
<td>1.572</td>
<td>10</td>
</tr>
<tr>
<td>401</td>
<td>TRAP</td>
<td>0</td>
<td>-0.598</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>402</td>
<td>TRAP</td>
<td>0</td>
<td>-1.676</td>
<td>2.913</td>
<td>10</td>
</tr>
<tr>
<td>402</td>
<td>UNI</td>
<td>-1.676</td>
<td>2.915</td>
<td>3.543</td>
<td>10</td>
</tr>
<tr>
<td>402</td>
<td>TRAP</td>
<td>-1.676</td>
<td>0</td>
<td>3.545</td>
<td>10</td>
</tr>
<tr>
<td>402</td>
<td>TRAP</td>
<td>-0.989</td>
<td>-0.49</td>
<td>3.921</td>
<td>10</td>
</tr>
<tr>
<td>402</td>
<td>TRAP</td>
<td>0</td>
<td>-0.989</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>403</td>
<td>TRAP</td>
<td>0</td>
<td>-0.49</td>
<td>0</td>
<td>2.489</td>
</tr>
<tr>
<td>403</td>
<td>TRAP</td>
<td>-0.569</td>
<td>0</td>
<td>1.498</td>
<td>3.417</td>
</tr>
<tr>
<td>404</td>
<td>TRAP</td>
<td>0</td>
<td>-0.569</td>
<td>1</td>
<td>4.96</td>
</tr>
<tr>
<td>405</td>
<td>TRAP</td>
<td>-0.237</td>
<td>0</td>
<td>3.779</td>
<td>6.007</td>
</tr>
<tr>
<td>405</td>
<td>TRAP</td>
<td>0</td>
<td>-0.237</td>
<td>0</td>
<td>3.778</td>
</tr>
<tr>
<td>406</td>
<td>TRAP</td>
<td>-0.94</td>
<td>0</td>
<td>4.815</td>
<td>9.416</td>
</tr>
<tr>
<td>406</td>
<td>TRAP</td>
<td>0</td>
<td>-0.94</td>
<td>0</td>
<td>4.813</td>
</tr>
<tr>
<td>407</td>
<td>TRAP</td>
<td>-1.87</td>
<td>0</td>
<td>2.629</td>
<td>6.102</td>
</tr>
<tr>
<td>407</td>
<td>TRAP</td>
<td>0</td>
<td>-1.87</td>
<td>0</td>
<td>2.628</td>
</tr>
<tr>
<td>408</td>
<td>TRAP</td>
<td>-0.926</td>
<td>0</td>
<td>4.8</td>
<td>9.365</td>
</tr>
<tr>
<td>408</td>
<td>TRAP</td>
<td>0</td>
<td>-0.926</td>
<td>0</td>
<td>4.799</td>
</tr>
<tr>
<td>409</td>
<td>TRAP</td>
<td>-1.138</td>
<td>0</td>
<td>0</td>
<td>1.624</td>
</tr>
<tr>
<td>409</td>
<td>TRAP</td>
<td>-0.472</td>
<td>0</td>
<td>0.445</td>
<td>1.623</td>
</tr>
<tr>
<td>409</td>
<td>UNI</td>
<td>-0.472</td>
<td>0</td>
<td>0.443</td>
<td>10</td>
</tr>
<tr>
<td>410</td>
<td>TRAP</td>
<td>-1.111</td>
<td>-2.09</td>
<td>0</td>
<td>1.038</td>
</tr>
<tr>
<td>410</td>
<td>TRAP</td>
<td>-2.09</td>
<td>-1.976</td>
<td>1.04</td>
<td>1.178</td>
</tr>
<tr>
<td>410</td>
<td>TRAP</td>
<td>-0.472</td>
<td>0</td>
<td>0</td>
<td>1.178</td>
</tr>
<tr>
<td>411</td>
<td>TRAP</td>
<td>-0.472</td>
<td>0</td>
<td>0.445</td>
<td>1.623</td>
</tr>
<tr>
<td>411</td>
<td>UNI</td>
<td>-0.472</td>
<td>0</td>
<td>0.443</td>
<td>10</td>
</tr>
<tr>
<td>411</td>
<td>TRAP</td>
<td>-0.58</td>
<td>0</td>
<td>0</td>
<td>1.624</td>
</tr>
</tbody>
</table>

LOAD 10 LOADTYPE Live TITLE LIVE LOAD
JOINT LOAD
Application Examples
EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th></th>
<th>FY</th>
<th>MX</th>
<th>MZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>92</td>
<td>-2.039E-14</td>
<td>-1.210336E-21</td>
<td>-3.006417E-21</td>
</tr>
<tr>
<td>103</td>
<td>-2.67466E-14</td>
<td>4.011332E-21</td>
<td>-1.407982E-21</td>
</tr>
<tr>
<td>119</td>
<td>-1.951477E-15</td>
<td>8.142299E-38</td>
<td>-6.64891E-22</td>
</tr>
<tr>
<td>134</td>
<td>-2.039E-14</td>
<td>-1.210336E-21</td>
<td>-3.006417E-21</td>
</tr>
<tr>
<td>145</td>
<td>-2.67466E-14</td>
<td>4.011332E-21</td>
<td>-1.407982E-21</td>
</tr>
</tbody>
</table>

MEMBER LOAD

<table>
<thead>
<tr>
<th></th>
<th>TRAP</th>
<th>UNI</th>
</tr>
</thead>
<tbody>
<tr>
<td>217</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.571 0 2.8535</td>
</tr>
<tr>
<td>218</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.571 0 2.8535</td>
</tr>
<tr>
<td>219</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.571 0 2.8535</td>
</tr>
<tr>
<td>220</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.571 0 2.8535</td>
</tr>
<tr>
<td>221</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>222</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>223</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>224</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>225</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>226</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>227</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>228</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td>Y 0 -0.8 0 3.9985</td>
</tr>
</tbody>
</table>

STAAD.Pro 5020 User Manual
Application Examples
EX. Interactive Concrete Design Examples

<p>| | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>228</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>229</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>229</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>229</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>229</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>230</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.404</td>
<td>0 1.179</td>
</tr>
<tr>
<td>230</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
<td>0 1.1785</td>
</tr>
<tr>
<td>231</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.719</td>
<td>0</td>
<td>0 2.4285</td>
</tr>
<tr>
<td>231</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.344</td>
<td>0</td>
<td>1.307 2.4285</td>
</tr>
<tr>
<td>231</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.344</td>
<td>0 1.3057</td>
</tr>
<tr>
<td>232</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
<td>0 1.1785</td>
</tr>
<tr>
<td>232</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.236</td>
<td>0</td>
<td>1.18 2.3585</td>
</tr>
<tr>
<td>232</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.453</td>
<td>0 2.3585</td>
</tr>
<tr>
<td>233</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.386</td>
<td>0 1.6032</td>
</tr>
<tr>
<td>233</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.306</td>
<td>0</td>
<td>1.605 3.4885</td>
</tr>
<tr>
<td>233</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.607</td>
<td>0</td>
<td>0.803 3.4885</td>
</tr>
<tr>
<td>233</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.453</td>
<td>-0.607</td>
<td>0 0.8012</td>
</tr>
<tr>
<td>234</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>234</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>234</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>235</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>236</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>236</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>236</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>236</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>237</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>238</td>
<td>UNI</td>
<td>Y</td>
<td>-0.249</td>
<td>0</td>
<td>5.8285</td>
</tr>
<tr>
<td>238</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.553</td>
<td>0 3.5743</td>
</tr>
<tr>
<td>238</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.553</td>
<td>-0.254</td>
<td>3.576 5.8285</td>
</tr>
<tr>
<td>239</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.024</td>
<td>0</td>
<td>0.988 1.9185</td>
</tr>
<tr>
<td>239</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.024</td>
<td>0 0.9863</td>
</tr>
<tr>
<td>239</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.254</td>
<td>0</td>
<td>0 1.9185</td>
</tr>
<tr>
<td>240</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.583</td>
<td>0 2.9135</td>
</tr>
<tr>
<td>240</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583</td>
<td>0</td>
<td>2.915 5.8285</td>
</tr>
<tr>
<td>240</td>
<td>UNI</td>
<td>Y</td>
<td>-0.249</td>
<td>0</td>
<td>5.8285</td>
</tr>
<tr>
<td>241</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>241</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>242</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>242</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>243</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>243</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>244</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>244</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4 7.9985</td>
</tr>
<tr>
<td>245</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.583</td>
<td>0 2.9135</td>
</tr>
<tr>
<td>245</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583</td>
<td>0</td>
<td>2.915 5.8285</td>
</tr>
<tr>
<td>246</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.766</td>
<td>0 2.2466</td>
</tr>
<tr>
<td>246</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.766</td>
<td>0</td>
<td>2.248 4.8485</td>
</tr>
<tr>
<td>247</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
<tr>
<td>247</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
</tr>
<tr>
<td>248</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.95 8.9485</td>
</tr>
<tr>
<td>248</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>0 3.9985</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Node</th>
<th>Type</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Node</th>
</tr>
</thead>
<tbody>
<tr>
<td>248</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
<td></td>
</tr>
<tr>
<td>248</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>5.05</td>
<td>9.0485</td>
</tr>
<tr>
<td>249</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>249</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
<td></td>
</tr>
<tr>
<td>249</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.9</td>
<td>8.8985</td>
</tr>
<tr>
<td>250</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.855</td>
<td>5.7085</td>
</tr>
<tr>
<td>250</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>251</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
<td></td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.95</td>
<td>8.9485</td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>251</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
<td></td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.95</td>
<td>8.9485</td>
</tr>
<tr>
<td>251</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>252</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
<td></td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>5.05</td>
<td>9.0485</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>252</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
<td></td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>5.05</td>
<td>9.0485</td>
</tr>
<tr>
<td>252</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>253</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
<td></td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>253</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9</td>
<td>8.9485</td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>253</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9</td>
<td>8.9485</td>
</tr>
<tr>
<td>253</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.855</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.855</td>
<td>5.7085</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>254</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>255</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
<td></td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.95</td>
<td>8.9485</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>255</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
<td></td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.95</td>
<td>8.9485</td>
</tr>
<tr>
<td>255</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>256</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
<td></td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>5.05</td>
<td>9.0485</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>256</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
<td></td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>5.05</td>
<td>9.0485</td>
</tr>
<tr>
<td>256</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>257</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
<td></td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.9</td>
<td>8.8985</td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>257</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
<td></td>
</tr>
<tr>
<td>257</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.9</td>
<td>8.8985</td>
</tr>
<tr>
<td>258</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.855</td>
<td>5.7085</td>
</tr>
<tr>
<td>258</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>258</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>258</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>259</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
<td></td>
</tr>
<tr>
<td>259</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.95</td>
<td>8.9485</td>
</tr>
<tr>
<td>259</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
<tr>
<td>259</td>
<td>UNI</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
<td></td>
</tr>
<tr>
<td>259</td>
<td>TRAP</td>
<td>-0.8</td>
<td>0</td>
<td>4.95</td>
<td>8.9485</td>
</tr>
<tr>
<td>260</td>
<td>TRAP</td>
<td>0</td>
<td>-0.8</td>
<td>0</td>
<td>3.9985</td>
</tr>
</tbody>
</table>
Application Examples
EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>260</th>
<th>UNI</th>
<th>Y</th>
<th>-0.8</th>
<th>4</th>
<th>5.0485</th>
</tr>
</thead>
<tbody>
<tr>
<td>260</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>5.05</td>
</tr>
<tr>
<td>260</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>260</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
</tr>
<tr>
<td>260</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>5.05</td>
</tr>
<tr>
<td>261</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>261</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
</tr>
<tr>
<td>261</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.9</td>
</tr>
<tr>
<td>261</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>261</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
</tr>
<tr>
<td>261</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.9</td>
</tr>
<tr>
<td>262</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.571</td>
<td>0</td>
<td>2.855</td>
</tr>
<tr>
<td>262</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
</tr>
<tr>
<td>262</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.571</td>
<td>0</td>
<td>2.855</td>
</tr>
<tr>
<td>262</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
</tr>
<tr>
<td>263</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>263</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
</tr>
<tr>
<td>263</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>-0.498</td>
<td>4.95</td>
</tr>
<tr>
<td>263</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.583</td>
<td>0</td>
</tr>
<tr>
<td>263</td>
<td>UNI</td>
<td>Y</td>
<td>-0.583</td>
<td>2.915</td>
<td>3.5435</td>
</tr>
<tr>
<td>263</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583</td>
<td>0</td>
<td>3.545</td>
</tr>
<tr>
<td>264</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.498</td>
<td>0</td>
<td>2.4885</td>
</tr>
<tr>
<td>265</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>265</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
</tr>
<tr>
<td>265</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>-0.65</td>
<td>5.05</td>
</tr>
<tr>
<td>265</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.684</td>
<td>0</td>
<td>2.844</td>
</tr>
<tr>
<td>265</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.684</td>
<td>0</td>
</tr>
<tr>
<td>266</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.65</td>
<td>-0.325</td>
<td>0</td>
</tr>
<tr>
<td>266</td>
<td>UNI</td>
<td>Y</td>
<td>-0.236</td>
<td>1.18</td>
<td>1.6235</td>
</tr>
<tr>
<td>266</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
<td>0</td>
</tr>
<tr>
<td>267</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>267</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
</tr>
<tr>
<td>267</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.9</td>
</tr>
<tr>
<td>267</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.354</td>
<td>0</td>
<td>3.89</td>
</tr>
<tr>
<td>267</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.354</td>
<td>0</td>
</tr>
<tr>
<td>268</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.571</td>
<td>0</td>
<td>2.855</td>
</tr>
<tr>
<td>268</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
</tr>
<tr>
<td>268</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.125</td>
<td>0</td>
<td>1.952</td>
</tr>
<tr>
<td>268</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.125</td>
<td>0</td>
</tr>
<tr>
<td>269</td>
<td>UNI</td>
<td>Y</td>
<td>-0.236</td>
<td>1.18</td>
<td>1.6235</td>
</tr>
<tr>
<td>269</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
<td>0</td>
</tr>
<tr>
<td>269</td>
<td>UNI</td>
<td>Y</td>
<td>-0.299</td>
<td>1.572</td>
<td>1.6235</td>
</tr>
<tr>
<td>269</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.299</td>
<td>0</td>
</tr>
<tr>
<td>270</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.583</td>
<td>0</td>
</tr>
<tr>
<td>270</td>
<td>UNI</td>
<td>Y</td>
<td>-0.583</td>
<td>2.915</td>
<td>3.5435</td>
</tr>
<tr>
<td>270</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583</td>
<td>0</td>
<td>3.545</td>
</tr>
<tr>
<td>270</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.36</td>
<td>-0.178</td>
<td>3.921</td>
</tr>
<tr>
<td>270</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.36</td>
<td>0</td>
</tr>
<tr>
<td>271</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.178</td>
<td>0</td>
<td>2.4885</td>
</tr>
<tr>
<td>272</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.928</td>
<td>0</td>
</tr>
<tr>
<td>272</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.928</td>
<td>0</td>
<td>4.842</td>
</tr>
<tr>
<td>273</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.918</td>
<td>0</td>
<td>5.373</td>
</tr>
<tr>
<td>273</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.918</td>
<td>0</td>
</tr>
<tr>
<td>274</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.284</td>
<td>0</td>
<td>1.498</td>
</tr>
<tr>
<td>274</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.284</td>
<td>0</td>
</tr>
<tr>
<td>274</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.751</td>
<td>0</td>
</tr>
<tr>
<td>275</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.119</td>
<td>0</td>
<td>3.779</td>
</tr>
<tr>
<td>275</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.119</td>
<td>0</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>276</td>
<td>TRAP</td>
<td>Y -0.342 0 4.815 9.4159</td>
<td></td>
</tr>
<tr>
<td>276</td>
<td>TRAP</td>
<td>Y 0 -0.342 0 4.8131</td>
<td></td>
</tr>
<tr>
<td>277</td>
<td>TRAP</td>
<td>Y -1 0 1.131 6.1018</td>
<td></td>
</tr>
<tr>
<td>277</td>
<td>TRAP</td>
<td>Y -0.751 -1 0 1.1295</td>
<td></td>
</tr>
<tr>
<td>277</td>
<td>TRAP</td>
<td>Y -0.65 0 2.629 6.1018</td>
<td></td>
</tr>
<tr>
<td>277</td>
<td>TRAP</td>
<td>Y 0 -0.65 0 2.6277</td>
<td></td>
</tr>
<tr>
<td>278</td>
<td>TRAP</td>
<td>Y -0.337 0 4.8 9.3653</td>
<td></td>
</tr>
<tr>
<td>278</td>
<td>TRAP</td>
<td>Y 0 -0.337 0 4.7989</td>
<td></td>
</tr>
<tr>
<td>279</td>
<td>TRAP</td>
<td>Y 0 -1.02 0 4.6612</td>
<td></td>
</tr>
<tr>
<td>279</td>
<td>TRAP</td>
<td>Y -1.02 0 4.663 8.9963</td>
<td></td>
</tr>
<tr>
<td>280</td>
<td>TRAP</td>
<td>Y -0.325 0 0 1.6235</td>
<td></td>
</tr>
<tr>
<td>280</td>
<td>TRAP</td>
<td>Y -0.236 0 0 1.6235</td>
<td></td>
</tr>
<tr>
<td>280</td>
<td>UNI</td>
<td>Y -0.236 0 0.4435</td>
<td></td>
</tr>
<tr>
<td>281</td>
<td>TRAP</td>
<td>Y -0.404 -0.76 0 1.0382</td>
<td></td>
</tr>
<tr>
<td>281</td>
<td>TRAP</td>
<td>Y -0.76 -0.719 1.04 1.178</td>
<td></td>
</tr>
<tr>
<td>281</td>
<td>TRAP</td>
<td>Y -0.236 0 0 1.178</td>
<td></td>
</tr>
<tr>
<td>282</td>
<td>TRAP</td>
<td>Y -0.236 0 0.445 1.6235</td>
<td></td>
</tr>
<tr>
<td>282</td>
<td>UNI</td>
<td>Y -0.236 0 0.4435</td>
<td></td>
</tr>
<tr>
<td>283</td>
<td>TRAP</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td></td>
</tr>
<tr>
<td>283</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>283</td>
<td>TRAP</td>
<td>Y 0 -0.571 0 2.8535</td>
<td></td>
</tr>
<tr>
<td>284</td>
<td>TRAP</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td></td>
</tr>
<tr>
<td>284</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>285</td>
<td>TRAP</td>
<td>Y 0 -0.571 0 2.8535</td>
<td></td>
</tr>
<tr>
<td>285</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>286</td>
<td>TRAP</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td></td>
</tr>
<tr>
<td>286</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>287</td>
<td>TRAP</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td></td>
</tr>
<tr>
<td>287</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>287</td>
<td>TRAP</td>
<td>Y 0 -0.571 0 2.8535</td>
<td></td>
</tr>
<tr>
<td>287</td>
<td>TRAP</td>
<td>Y 0 -0.8 0 3.9985</td>
<td></td>
</tr>
<tr>
<td>287</td>
<td>TRAP</td>
<td>Y -0.8 0 4 7.9985</td>
<td></td>
</tr>
<tr>
<td>288</td>
<td>TRAP</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td></td>
</tr>
<tr>
<td>288</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>288</td>
<td>TRAP</td>
<td>Y 0 -0.571 0 2.8535</td>
<td></td>
</tr>
<tr>
<td>288</td>
<td>TRAP</td>
<td>Y 0 -0.8 0 3.9985</td>
<td></td>
</tr>
<tr>
<td>288</td>
<td>TRAP</td>
<td>Y -0.8 0 4 7.9985</td>
<td></td>
</tr>
<tr>
<td>289</td>
<td>TRAP</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td></td>
</tr>
<tr>
<td>289</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>289</td>
<td>TRAP</td>
<td>Y 0 -0.571 0 2.8535</td>
<td></td>
</tr>
<tr>
<td>289</td>
<td>TRAP</td>
<td>Y 0 -0.8 0 3.9985</td>
<td></td>
</tr>
<tr>
<td>289</td>
<td>TRAP</td>
<td>Y -0.8 0 4 7.9985</td>
<td></td>
</tr>
<tr>
<td>290</td>
<td>TRAP</td>
<td>Y -0.571 0 5.145 7.9985</td>
<td></td>
</tr>
<tr>
<td>290</td>
<td>UNI</td>
<td>Y -0.571 2.855 5.1435</td>
<td></td>
</tr>
<tr>
<td>290</td>
<td>TRAP</td>
<td>Y 0 -0.571 0 2.8535</td>
<td></td>
</tr>
<tr>
<td>290</td>
<td>TRAP</td>
<td>Y 0 -0.8 0 3.9985</td>
<td></td>
</tr>
<tr>
<td>290</td>
<td>TRAP</td>
<td>Y -0.8 0 4 7.9985</td>
<td></td>
</tr>
<tr>
<td>290</td>
<td>TRAP</td>
<td>Y 0 -0.8 0 3.9985</td>
<td></td>
</tr>
</tbody>
</table>
Application Examples
EX. Interactive Concrete Design Examples

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>292</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>293</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>293</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>293</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>293</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>294</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>294</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>294</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>294</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>295</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>295</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>295</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>295</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>296</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.484</td>
</tr>
<tr>
<td>296</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
</tr>
<tr>
<td>297</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.719</td>
<td>0</td>
</tr>
<tr>
<td>297</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.344</td>
<td>0</td>
</tr>
<tr>
<td>298</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.344</td>
</tr>
<tr>
<td>298</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
</tr>
<tr>
<td>298</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.236</td>
<td>0</td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.453</td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.306</td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.306</td>
<td>0</td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.607</td>
<td>0</td>
</tr>
<tr>
<td>299</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.453</td>
<td>-0.607</td>
</tr>
<tr>
<td>300</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>300</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>300</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>300</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>301</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>301</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>301</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>301</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>302</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>302</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>302</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>302</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>303</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>303</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>303</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>303</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>304</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.249</td>
<td>0</td>
</tr>
<tr>
<td>304</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.553</td>
</tr>
<tr>
<td>304</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.553</td>
<td>-0.254</td>
</tr>
<tr>
<td>305</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.024</td>
<td>0</td>
</tr>
<tr>
<td>305</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.024</td>
</tr>
<tr>
<td>305</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.254</td>
<td>0</td>
</tr>
<tr>
<td>306</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.583</td>
</tr>
<tr>
<td>306</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583</td>
<td>0</td>
</tr>
<tr>
<td>306</td>
<td>UNI</td>
<td>Y</td>
<td>-0.249</td>
<td>0</td>
</tr>
<tr>
<td>307</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>307</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>308</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>308</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>309</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>309</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>310</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>310</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
</tbody>
</table>
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>TRAP</th>
<th>Y</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>311</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583 0 2.9135</td>
</tr>
<tr>
<td>311</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583 0 2.915 5.8285</td>
</tr>
<tr>
<td>312</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.766 0 2.2466</td>
</tr>
<tr>
<td>312</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.766 0 2.248 4.8485</td>
</tr>
<tr>
<td>313</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>313</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.9485</td>
</tr>
<tr>
<td>313</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.95 8.9485</td>
</tr>
<tr>
<td>314</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>314</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 5.0485</td>
</tr>
<tr>
<td>314</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 5.05 9.0485</td>
</tr>
<tr>
<td>315</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>315</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>315</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>316</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>316</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 2.855 5.7085</td>
</tr>
<tr>
<td>316</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 2.8535</td>
</tr>
<tr>
<td>317</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>317</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.9485</td>
</tr>
<tr>
<td>317</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.95 8.9485</td>
</tr>
<tr>
<td>317</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>317</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 5.0485</td>
</tr>
<tr>
<td>317</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 5.05 9.0485</td>
</tr>
<tr>
<td>318</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>318</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 5.0485</td>
</tr>
<tr>
<td>318</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 5.05 9.0485</td>
</tr>
<tr>
<td>318</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>318</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>318</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>319</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>319</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>319</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>320</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>320</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.571 0 2.855 5.7085</td>
</tr>
<tr>
<td>320</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.571 0 2.8535</td>
</tr>
<tr>
<td>320</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.571 0 2.855 5.7085</td>
</tr>
<tr>
<td>320</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.571 0 2.8535</td>
</tr>
<tr>
<td>321</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>321</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.9485</td>
</tr>
<tr>
<td>321</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>321</td>
<td>TRAP</td>
<td>Y</td>
<td>0 -0.8 0 3.9985</td>
</tr>
<tr>
<td>321</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.9485</td>
</tr>
<tr>
<td>321</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.95 8.9485</td>
</tr>
<tr>
<td>322</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>322</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.95 8.9485</td>
</tr>
<tr>
<td>322</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 5.05 9.0485</td>
</tr>
<tr>
<td>322</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>322</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 5.0485</td>
</tr>
<tr>
<td>322</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 5.05 9.0485</td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>323</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>323</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>323</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>324</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 5.05 9.0485</td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>324</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 3.9985</td>
</tr>
<tr>
<td>324</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8 4 4.8985</td>
</tr>
<tr>
<td>324</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8 0 4.9 8.9485</td>
</tr>
</tbody>
</table>

**STAAD.Pro**

5026

**User Manual**
<table>
<thead>
<tr>
<th>Step</th>
<th>Type</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Force</th>
</tr>
</thead>
<tbody>
<tr>
<td>338</td>
<td>TRAP</td>
<td>-0.928</td>
<td>0</td>
<td>4.842</td>
<td>10.0131</td>
</tr>
<tr>
<td>339</td>
<td>TRAP</td>
<td>-0.918</td>
<td>0</td>
<td>5.373</td>
<td>10.2531</td>
</tr>
<tr>
<td>340</td>
<td>TRAP</td>
<td>0</td>
<td>-0.918</td>
<td>0</td>
<td>5.3713</td>
</tr>
<tr>
<td>341</td>
<td>TRAP</td>
<td>-0.284</td>
<td>0</td>
<td>1.498</td>
<td>3.617</td>
</tr>
<tr>
<td>342</td>
<td>TRAP</td>
<td>0</td>
<td>-0.284</td>
<td>0</td>
<td>1.4963</td>
</tr>
<tr>
<td>343</td>
<td>TRAP</td>
<td>0</td>
<td>-0.751</td>
<td>0</td>
<td>3.417</td>
</tr>
<tr>
<td>344</td>
<td>TRAP</td>
<td>-0.119</td>
<td>0</td>
<td>3.779</td>
<td>6.0069</td>
</tr>
<tr>
<td>345</td>
<td>TRAP</td>
<td>0</td>
<td>-0.119</td>
<td>0</td>
<td>3.7777</td>
</tr>
<tr>
<td>346</td>
<td>TRAP</td>
<td>-0.342</td>
<td>0</td>
<td>4.815</td>
<td>9.4159</td>
</tr>
<tr>
<td>347</td>
<td>TRAP</td>
<td>0</td>
<td>-0.342</td>
<td>0</td>
<td>4.8131</td>
</tr>
<tr>
<td>348</td>
<td>TRAP</td>
<td>-1</td>
<td>0</td>
<td>1.131</td>
<td>6.1018</td>
</tr>
<tr>
<td>349</td>
<td>TRAP</td>
<td>-0.751</td>
<td>-1</td>
<td>0</td>
<td>1.1295</td>
</tr>
<tr>
<td>350</td>
<td>TRAP</td>
<td>-0.65</td>
<td>0</td>
<td>2.629</td>
<td>6.1018</td>
</tr>
<tr>
<td>351</td>
<td>TRAP</td>
<td>0</td>
<td>-0.65</td>
<td>0</td>
<td>2.6277</td>
</tr>
<tr>
<td>352</td>
<td>TRAP</td>
<td>-0.337</td>
<td>0</td>
<td>4.8</td>
<td>9.3653</td>
</tr>
<tr>
<td>353</td>
<td>TRAP</td>
<td>0</td>
<td>-0.337</td>
<td>0</td>
<td>4.7989</td>
</tr>
<tr>
<td>354</td>
<td>TRAP</td>
<td>0</td>
<td>-1</td>
<td>0</td>
<td>4.6612</td>
</tr>
<tr>
<td>355</td>
<td>TRAP</td>
<td>-1.02</td>
<td>0</td>
<td>4.663</td>
<td>8.9963</td>
</tr>
<tr>
<td>356</td>
<td>TRAP</td>
<td>-0.325</td>
<td>0</td>
<td>0</td>
<td>1.6235</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>0</td>
<td>-0.325</td>
<td>0</td>
<td>1.6235</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>0</td>
<td>-0.236</td>
<td>0</td>
<td>0.445</td>
</tr>
<tr>
<td>359</td>
<td>TRAP</td>
<td>0</td>
<td>-0.236</td>
<td>0</td>
<td>1.6235</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>7.9985</td>
</tr>
<tr>
<td>362</td>
<td>UNI</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>5.1435</td>
</tr>
<tr>
<td>363</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>364</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>7.9985</td>
</tr>
<tr>
<td>365</td>
<td>UNI</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>5.1435</td>
</tr>
<tr>
<td>366</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>367</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>7.9985</td>
</tr>
<tr>
<td>368</td>
<td>UNI</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>5.1435</td>
</tr>
<tr>
<td>369</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>370</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>7.9985</td>
</tr>
<tr>
<td>371</td>
<td>UNI</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>5.1435</td>
</tr>
<tr>
<td>372</td>
<td>TRAP</td>
<td>0</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
</tr>
<tr>
<td>373</td>
<td>TRAP</td>
<td>-0.571</td>
<td>0</td>
<td>2.8535</td>
<td>7.9985</td>
</tr>
</tbody>
</table>

Application Examples

EX. Interactive Concrete Design Examples
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>TRAP</th>
<th>Y</th>
<th>X1</th>
<th>X2</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>356</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.381</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.381</td>
<td>0</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.02</td>
<td>0</td>
</tr>
<tr>
<td>357</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.02</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>358</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>359</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>359</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>360</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>361</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>362</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.484</td>
</tr>
<tr>
<td>362</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
</tr>
<tr>
<td>363</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.719</td>
<td>0</td>
</tr>
<tr>
<td>363</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.344</td>
<td>0</td>
</tr>
<tr>
<td>363</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.344</td>
</tr>
<tr>
<td>364</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.236</td>
</tr>
<tr>
<td>364</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.236</td>
<td>0</td>
</tr>
<tr>
<td>364</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.453</td>
</tr>
<tr>
<td>365</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.396</td>
</tr>
<tr>
<td>365</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.396</td>
<td>0</td>
</tr>
<tr>
<td>365</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.607</td>
<td>0</td>
</tr>
<tr>
<td>365</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.453</td>
<td>-0.607</td>
</tr>
<tr>
<td>366</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>366</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>366</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>366</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>367</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>367</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>367</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>367</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>368</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>368</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>368</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>368</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>369</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>369</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>369</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
</tr>
<tr>
<td>369</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
</tr>
<tr>
<td>370</td>
<td>UNI</td>
<td>Y</td>
<td>-0.249</td>
<td>0</td>
</tr>
<tr>
<td>370</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.553</td>
</tr>
<tr>
<td>370</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.553</td>
<td>-0.254</td>
</tr>
<tr>
<td>371</td>
<td>TRAP</td>
<td>Y</td>
<td>-1.024</td>
<td>0</td>
</tr>
<tr>
<td>371</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-1.024</td>
</tr>
<tr>
<td>371</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.254</td>
<td>0</td>
</tr>
<tr>
<td>372</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.583</td>
</tr>
<tr>
<td>372</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583</td>
<td>0</td>
</tr>
<tr>
<td>372</td>
<td>UNI</td>
<td>Y</td>
<td>-0.249</td>
<td>0</td>
</tr>
</tbody>
</table>
### Application Examples

#### EX. Interactive Concrete Design Examples

<table>
<thead>
<tr>
<th>Line</th>
<th>Action</th>
<th>Number</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>373</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>373</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>7.9985</td>
</tr>
<tr>
<td>374</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>374</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>7.9985</td>
</tr>
<tr>
<td>375</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>375</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>7.9985</td>
</tr>
<tr>
<td>376</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>376</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>7.9985</td>
</tr>
<tr>
<td>377</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>2.9135</td>
</tr>
<tr>
<td>377</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.583</td>
<td>0</td>
<td>2.9155.8285</td>
</tr>
<tr>
<td>378</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.766</td>
<td>2.2466</td>
</tr>
<tr>
<td>378</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.766</td>
<td>0</td>
<td>2.2484.8485</td>
</tr>
<tr>
<td>379</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>379</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
</tr>
<tr>
<td>379</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.958.9485</td>
</tr>
<tr>
<td>380</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>380</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
</tr>
<tr>
<td>380</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>5.059.0485</td>
</tr>
<tr>
<td>381</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>381</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
</tr>
<tr>
<td>381</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.98.8985</td>
</tr>
<tr>
<td>382</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.571</td>
<td>0</td>
<td>2.8555.7085</td>
</tr>
<tr>
<td>382</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.571</td>
<td>2.8535</td>
</tr>
<tr>
<td>383</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>383</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
</tr>
<tr>
<td>383</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.958.9485</td>
</tr>
<tr>
<td>384</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>384</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
</tr>
<tr>
<td>384</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>5.059.0485</td>
</tr>
<tr>
<td>385</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>385</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.8985</td>
</tr>
<tr>
<td>385</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.98.8985</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.571</td>
<td>0</td>
<td>2.8555.7085</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.571</td>
<td>2.8535</td>
</tr>
<tr>
<td>386</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.571</td>
<td>0</td>
<td>2.8555.7085</td>
</tr>
<tr>
<td>387</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>387</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>4.9485</td>
</tr>
<tr>
<td>387</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>4.958.9485</td>
</tr>
<tr>
<td>388</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
<tr>
<td>388</td>
<td>UNI</td>
<td>Y</td>
<td>-0.8</td>
<td>4</td>
<td>5.0485</td>
</tr>
<tr>
<td>388</td>
<td>TRAP</td>
<td>Y</td>
<td>-0.8</td>
<td>0</td>
<td>5.059.0485</td>
</tr>
<tr>
<td>389</td>
<td>TRAP</td>
<td>Y</td>
<td>0</td>
<td>-0.8</td>
<td>3.9985</td>
</tr>
</tbody>
</table>
Application Examples

EX. Interactive Concrete Design Examples

STAAD.Pro 5031 User Manual
### Application Examples

**EX. Interactive Concrete Design Examples**

<table>
<thead>
<tr>
<th>LOAD COMB</th>
<th>Forces/Directives</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>1.5D+1.5L</td>
</tr>
<tr>
<td>20</td>
<td>1.2D+1.2L+1.2WX</td>
</tr>
<tr>
<td>21</td>
<td>1.2D+1.2L-1.2WX</td>
</tr>
<tr>
<td>22</td>
<td>1.2D+1.2L+1.2WZ</td>
</tr>
<tr>
<td>23</td>
<td>1.2D+1.2L-1.2WZ</td>
</tr>
<tr>
<td>36</td>
<td>1.5D+1.5WX</td>
</tr>
<tr>
<td>37</td>
<td>1.5D-1.5WX</td>
</tr>
<tr>
<td>38</td>
<td>1.5D+1.5WZ</td>
</tr>
<tr>
<td>39</td>
<td>1.5D-1.5WZ</td>
</tr>
<tr>
<td>52</td>
<td>0.9D+1.5WX</td>
</tr>
<tr>
<td>53</td>
<td>0.9D-1.5WX</td>
</tr>
<tr>
<td>54</td>
<td>0.9D+1.5WZ</td>
</tr>
<tr>
<td>55</td>
<td>0.9D-1.5WZ</td>
</tr>
</tbody>
</table>

**PERFORM ANALYSIS**

**PRINT MEMBER INFORMATION**

**UNIT METERS MTONS**

**PRINT MEMBER FORCES**

**PRINT ELEMENT STRESS**

**PRINT SUPPORT REACTIONS**
EX. Bride Deck Loading Example

This example demonstrates how to apply vehicle loads per BS 5400 Part 2 on two lanes of a two span, steel girder bridge using the Bridge Deck workflow.

EX. To open the model in Bridge Deck workflow

1. Create a new file in STAAD.Pro.
2. Open the file and copy/paste the input file data (on page 5036).
3. Run an analysis on the structure.
4. Select Bridge Deck in the Workflows panel.

   Note: If you do not have a license for this module, you will not be able to proceed.

The Bridge Deck ribbon tab opens.

EX. To define the bridge deck

1. (Optional) If the mouse pointer is not already displayed as the Plate Cursor, in the Analytical Modeling workflow on the Geometry ribbon tab, select the Plates Cursor tool in the Selection group.

2. Press <CTRL+A>.
   All plate elements are selected in the entire model.
3. In the Bridge Deck workflow on the Bridge Deck ribbon tab, select the Create Deck tool in the Deck group.

   The Save Deck as dialog opens.
4. Type Northbound for the Name.
5. Click OK.

EX. To generate the influence surface for the deck

1. On the Bridge Deck ribbon tab, select the Loading > Influence Surface Generator tool in the Loading group.
A temporary STAAD input file (filename_deck.std) is sent to the STAAD analysis engine for processing. The STAAD Analysis and Design dialog opens to display the progress. When the analysis is complete, the dialog closes automatically.

2. On the Bridge Deck ribbon tab, select the Loading > Influence Diagram tool in the Loading group. The Diagrams dialog Influence tab (on page 3136) opens.

3. Select the Diagram Type as Node Displacement.
   The Parameter field updates to display Node Displacement and the object field updates to Node.

4. Select the Node Displacement as Y (Global Y direction).

5. Select the Node as 104 (approximately center of the deck, mid-span of the longer span).

6. Click OK.
   The influence surface for the vertical displacement at node 104 is drawn on the deck.

EX. To define the roadway lanes

**Note:** The deck is selected in the Bridge Deck ribbon tab Deck group by default, since only one deck has been defined. Otherwise, it would be necessary to select the correct deck on which a new roadway is to be defined.

1. On the Bridge Deck ribbon tab, select the Define Roadway tool in the Deck group.

   The Roadways dialog (on page 3123) opens.

2. Click New.
   The Define Roadway dialog (on page 3125) opens.

3. Type 0.61 m for Curb A: Origin Z value and 6.35 m for Curb B: Origin Z value. Leave all other values as the default.
   The preview updates to display the roadway on the currently selected deck.

4. Click OK.
   Roadway CW 1 is now added to the Roadways list in the Roadways dialog and is outlined in blue on the deck in the active view window.

5. Click Close.

EX. To place automatically generated loads on the roadway

1. On the Bridge Deck ribbon tab, select the Loading > Run Load Generator tool in the Loading group.
   The Load Generator Parameters dialog (on page 3130) opens.

2. On the General tab, Select BS5400 Part 2 as the Design Code.
   The <code>tab updates to BS5400.


4. Select the BS5400 tab (on page 3132).
   **Note:** For this example, the default single load case with the nominal HB vehicle and unit of 30 are OK.

5. Select the Node Displacements tab.
**Note**: An action must be defined for which the maximum results determine the placement of the moving loads. This example will use the same action as reviewed above in the influence surface.

6. Enter the following data into the Node Displacements table:

<table>
<thead>
<tr>
<th>Node</th>
<th>Displacement</th>
<th>Effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>104</td>
<td>Y</td>
<td>+ve</td>
</tr>
<tr>
<td>104</td>
<td>Y</td>
<td>-ve</td>
</tr>
</tbody>
</table>

7. Click **OK**. The program analyzes the deck to obtain the critical load positions for the specified action.

A summary of the analysis is opened in your default text editor (e.g., Notepad). The file (named `filename_deckx.out`) is saved in the same location as the input file. Close the text editor once you have completed reviewing this file.

**EX. To review the generated loads graphically**

1. Select **2:N04: Disp Y -ve** in the current load case drop-down list, found in the View toolbar.
2. Either:
   - Select View > Structure Diagrams....
   - or
   - Right-click anywhere in the active view window and then select Structure Diagrams... from the pop-up menu.
   - The **Diagrams** dialog opens.
3. Select the **Deck tab** (on page 3137).
4. Select to display **Loads** and **Vehicles** in addition to Roadways (on by default).
5. Click **OK**.

The placement of the loads and vehicles are overlayed on the bridge deck. Note that the defined roadway was wide enough for two lanes of traffic, and these were created automatically.

**EX. To transfer the load case to the STAAD.Pro model**

To transfer the generated load cases to STAAD.Pro for analysis and design, do the following.

1. On the **Bridge Deck** ribbon tab, select the **Loading > Create Loading in STAAD Model** tool in the **Loading** group.
   - A message dialog opens to confirm the load was successfully added in the STAAD input file.
2. Select the **Analytical Modeling** workflow.
3. On the **Loading** page, review the loads added to the input file in the **Load & Definitions** dialog.

   **Note**: One load was added for each action requested in the **Load Generators Parameters** dialog.

4. (Optional) Select the **Design | Steel** page and define parameters for the BS5400 code to perform a member design in addition to analysis for the generated loads.
5. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.
Application Examples
EX. Bride Deck Loading Example

EX. Bridge Deck Loading Input File
The following input is used for this example:
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Mar-06
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 1 0 1; 4 0 0 1; 5 2 0 0; 6 2 0 1; 7 3 0 0; 8 3 0 1;
9 4 0 0; 10 4 0 1; 11 5 0 0; 12 5 0 1; 13 6 0 0; 14 6 0 1; 15 7 0 0; 16 7 0 1;
17 8 0 0; 18 8 0 1; 19 9 0 0; 20 9 0 1; 21 10 0 0; 22 10 0 1; 23 11 0 0;
24 11 0 1; 25 12 0 0; 26 12 0 1; 27 13 0 0; 28 13 0 1; 29 14 0 0; 30 14 0 1;
31 15 0 0; 32 15 0 1; 33 16 0 0; 34 16 0 1; 35 17 0 0; 36 17 0 1; 37 18 0 0;
38 18 0 1; 39 19 0 0; 40 19 0 1; 41 20 0 0; 42 20 0 1; 43 21 0 0; 44 21 0 1;
45 22 0 0; 46 22 0 1; 47 23 0 0; 48 23 0 1; 49 24 0 0; 50 24 0 1; 51 25 0 0;
52 25 0 1; 53 26 0 0; 54 26 0 1; 55 27 0 0; 56 27 0 1; 57 28 0 0; 58 28 0 1;
59 29 0 0; 60 29 0 1; 61 30 0 0; 62 30 0 1; 63 1 0 2; 64 0 0 2; 65 2 0 2;
66 3 0 2; 67 4 0 2; 68 5 0 2; 69 6 0 2; 70 7 0 2; 71 8 0 2; 72 9 0 2;
73 10 0 2; 74 11 0 2; 75 12 0 2; 76 13 0 2; 77 14 0 2; 78 15 0 2; 79 16 0 2;
80 17 0 2; 81 18 0 2; 82 19 0 2; 83 20 0 2; 84 21 0 2; 85 22 0 2; 86 23 0 2;
87 24 0 2; 88 25 0 2; 89 26 0 2; 90 27 0 2; 91 28 0 2; 92 29 0 2; 93 30 0 2;
94 1 0 3; 95 0 0 3; 96 2 0 3; 97 3 0 3; 98 4 0 3; 99 5 0 3; 100 6 0 3;
101 7 0 3; 102 8 0 3; 103 9 0 3; 104 10 0 3; 105 11 0 3; 106 12 0 3;
107 13 0 3; 108 14 0 3; 109 15 0 3; 110 16 0 3; 111 17 0 3; 112 18 0 3;
113 19 0 3; 114 20 0 3; 115 21 0 3; 116 22 0 3; 117 23 0 3; 118 24 0 3;
119 25 0 3; 120 26 0 3; 121 27 0 3; 122 28 0 3; 123 29 0 3; 124 30 0 3;
125 1 0 4; 126 0 0 4; 127 2 0 4; 128 3 0 4; 129 4 0 4; 130 5 0 4; 131 6 0 4;
132 7 0 4; 133 8 0 4; 134 9 0 4; 135 10 0 4; 136 11 0 4; 137 12 0 4;
138 13 0 4; 139 14 0 4; 140 15 0 4; 141 16 0 4; 142 17 0 4; 143 18 0 4;
144 19 0 4; 145 20 0 4; 146 21 0 4; 147 22 0 4; 148 23 0 4; 149 24 0 4;
150 25 0 4; 151 26 0 4; 152 27 0 4; 153 28 0 4; 154 29 0 4; 155 30 0 4;
156 1 0 5; 157 0 0 5; 158 2 0 5; 159 3 0 5; 160 4 0 5; 161 5 0 5; 162 6 0 5;
163 7 0 5; 164 8 0 5; 165 9 0 5; 166 10 0 5; 167 11 0 5; 168 12 0 5;
169 13 0 5; 170 14 0 5; 171 15 0 5; 172 16 0 5; 173 17 0 5; 174 18 0 5;
175 19 0 5; 176 20 0 5; 177 21 0 5; 178 22 0 5; 179 23 0 5; 180 24 0 5;
181 25 0 5; 182 26 0 5; 183 27 0 5; 184 28 0 5; 185 29 0 5; 186 30 0 5;
187 1 0 6; 188 0 0 6; 189 2 0 6; 190 3 0 6; 191 4 0 6; 192 5 0 6; 193 6 0 6;
194 7 0 6; 195 8 0 6; 196 9 0 6; 197 10 0 6; 198 11 0 6; 199 12 0 6;
200 13 0 6; 201 14 0 6; 202 15 0 6; 203 16 0 6; 204 17 0 6; 205 18 0 6;
206 19 0 6; 207 20 0 6; 208 21 0 6; 209 22 0 6; 210 23 0 6; 211 24 0 6;
212 25 0 6; 213 26 0 6; 214 27 0 6; 215 28 0 6; 216 29 0 6; 217 30 0 6;
218 1 0 7; 219 0 0 7; 220 2 0 7; 221 3 0 7; 222 4 0 7; 223 5 0 7; 224 6 0 7;
225 7 0 7; 226 8 0 7; 227 9 0 7; 228 10 0 7; 229 11 0 7; 230 12 0 7;
231 13 0 7; 232 14 0 7; 233 15 0 7; 234 16 0 7; 235 17 0 7; 236 18 0 7;
237 19 0 7; 238 20 0 7; 239 21 0 7; 240 22 0 7; 241 23 0 7; 242 24 0 7;
243 25 0 7; 244 26 0 7; 245 27 0 7; 246 28 0 7; 247 29 0 7; 248 30 0 7;
249 0 -5 0; 250 0 -5 1; 251 0 -5 2; 252 0 -5 3; 253 0 -5 4; 254 0 -5 5;
255 0 -5 6; 256 0 -5 7; 257 20 -5 0; 258 20 -5 1; 259 20 -5 2; 260 20 -5 3;
261 20 -5 4; 262 20 -5 5; 263 20 -5 6; 264 20 -5 7; 265 30 -5 0; 266 30 -5 1;
267 30 -5 2; 268 30 -5 3; 269 30 -5 4; 270 30 -5 5; 271 30 -5 6; 272 30 -5 7;
MEMBER INCIDENCES
986 1 2; 989 41 43; 990 43 45; 991 45 47; 992 47 49; 993 49 51; 994 51 53;

STAAD.Pro

5036

User Manual


Application Examples

EX. Bride Deck Loading Example

STAAD.Pro 5038 User Manual
EX. To create model used for pushover example

To quickly set up the model used for this example, use the following procedure.
Only one pushover definition can be added to a model. Therefore, this example will start with a model you can add the pushover definition to. The end result will be functionally equivalent to the example model installed with the program.

1. Start an empty STAAD.Pro model in the analytical modeling workflow.
   Refer to GS. To create a new STAAD.Pro model (on page 33) for additional details.
2. On the Utilities ribbon tab, select the Command File tool in the Edit group.

   The STAAD.Pro Editor window opens.

3. Click Copy to Clipboard below.

   STAAD PLANE
   START JOB INFORMATION
   ENGINEER DATE 07-Oct-05
   END JOB INFORMATION
   INPUT WIDTH 79
   UNIT INCHES KIP
   JOINT COORDINATES
   1 0 0 0; 2 0 118.11 0; 3 118.11 118.11 0; 4 118.11 0 0; 5 0 236.22 0;
   6 118.11 236.22 0; 7 0 354.331 0; 8 118.11 354.331 0; 9 0 472.441 0;
   10 118.11 472.441 0; 11 236.22 118.11 0; 12 236.22 0 0; 13 236.22 236.22 0;
   14 236.22 354.331 0; 15 236.22 472.441 0;
   MEMBER INCIDENCES
   1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 5 7; 8 7 8; 9 8 6; 10 7 9; 11 9 10;
   12 10 8; 13 3 11; 14 6 13; 15 8 14; 16 10 15; 17 11 12; 18 13 11; 19 14 13;
   20 15 14;
   DEFINE MATERIAL START
   ISOTROPIC STEEL
   E 29732.7
   POISSON 0.3
   DENSITY 0.000283
   ALPHA 1.2e-005
   DAMP 0.03
   END DEFINE MATERIAL
   MEMBER PROPERTY AMERICAN
   2 5 13 14 TABLE ST W16X26
   8 11 15 16 TABLE ST W16X26
   1 3 4 6 17 18 TABLE ST W24X55
   7 9 10 12 19 20 TABLE ST W24X55
   CONSTANTS
   MATERIAL STEEL MEMB 1 TO 20
   SUPPORTS
   1 4 12 FIXED
   LOAD 1 LOADTYPE GRAVITY
   SELFWEIGHT Y -1
   MEMBER LOAD
   2 5 8 11 13 TO 16 UNI GY -0.2
   FINISH

4. In the STAAD.Pro Editor window, select the Paste tool on the Home ribbon tab.
The model contents are pasted into the editor.

5. Save and close the STAAD.Pro Editor.

The model opens in the Analytical Workflow.

EX. To define general pushover data

1. On the Loading ribbon tab, select the Pushover tool in the Define Loading Systems group. The Create New Definitions / Load Cases / Load Items dialog opens with only the Pushover tab displayed.

2. On the Define Input tab, select the General Input Parameters option.

3. Select Moment Frame as the Type of Frame.

4. Select Ignore Effect from the Geometric Non-Linearity Effect drop-down list.

5. Set the Displacement Incremental Value option and then type 0.010 (in) in the text field.

6. Click Add.

7. Select the Member Specific Parameter option.

8. Set the Expected Yield Stress option and then type 36.0 (kip/in²) in the text field.

Note: You will assign this parameter to members in a later procedure.

9. Click Add.

In the next procedure, you will define the loading pattern and spectrum data, so you can leave the Define Pushover dialog open.

EX. To define loading pattern and spectrum data

1. On the Define Pushover dialog, select the Define Loading Pattern tab.

2. Type 250 in the Number of Push Load Steps.

3. Click Add.

4. Select the Define Spectrum Data tab.

5. Type 5.0 (%) in the 1st Spectrum text field.

6. Delete the values from the remaining spectrum fields.

7. Select Class A from the Site Category drop-down list.

8. Type 1.0 in both the At Short Period and At One - Second Period text fields.

9. Click Add.

In the next procedure, you will define the solution control, so you can leave the Define Pushover dialog open.

EX. To define the solution control

1. On the Define Pushover dialog, select the Define Solution Control tab.

2. Select the Push Upto Defined Displacement at Control Joint option.

3. Select X Axis for the Direction.

4. Type 5.00 (in) in the Joint Displacement Value text field.
5. Select 10 as the Joint Number.
   The drop-down list contains the node numbers within the entire model.
6. Click Add.
7. Click Close.

EX. To assign the member-specific parameters

1. On the Load & Definition dialog, expand the Definitions > Pushover Definitions section.
   You will see the input parameters for the pushover definition you have added.
2. Select the entry labeled FYE 36.00000.
3. In the Assignment Method group, select Assign To View.
4. Click Assign.
   You are prompted to confirm you want to assign to all visible members in the view.
5. Click Yes.

EX. To specify and run the pushover analysis

1. Either:
   select the Analysis page
   or
   on the Analysis and Design ribbon tab, select the Analysis Commands tool in the Analysis Data group

   The Analysis/Print Commands dialog opens.
2. Select the Perform Pushover Analysis tab.
3. Click Add.
4. Click Close.
5. On the Analysis and Design ribbon tab, select the Run Analysis tool in the Analysis group.

   The STAAD Analysis and Design dialog opens.
6. Select the Go to Post Processing Mode option.
7. Click Done.

   The Postprocessing Workflow opens.

EX. To review pushover displacement results

If you did not select the Postprocessing option in the results dialog in the previous procedure, select the Postprocessing workflow.
1. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
2. Select the **Graphs** tool in the **Pushover** group.
   The Capacity Curve graph and Capacity Curve table open.
   
   **Note:** The last load step (63) has a displacement that exceeds the maximum of 5.0 in specified in the pushover parameters.

3. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
4. Select the **Node Results** tool in the **Pushover** group.
   The Node Displacements and Support Reactions tables open.
   
   **Note:** These tables display the values for the entire structure for a selected load step. The deflected shape for the current load step is also displayed graphically.

5. Type 63 in the **Select Load Step** field and then press `<Enter>`.
   The tables and view window update.

6. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
7. Select the **Beam Results** tool in the **Pushover** group.
   The Beam Hinge Results and Beam Force Detail tables open.

8. Repeat clicking the up arrow adjacent to either **Select Load Step** field until load step 4 is displayed.
   
   **Note:** The first hinge initially forms at this step. This is denoted with a green hinge, representing immediate occupancy status.

9. Repeat clicking the up arrow until about load step 44 is reached.
   Several more hinges have formed at the top of the elastic range and the first hinge has now turned blue, representing the Life Safety range of the acceptance criteria.
OpenSTAAD is a library of exposed functions allowing engineers access to STAAD.Pro's internal functions and routines as well as its graphical commands. With OpenSTAAD, you can use Visual Basic for Applications (VBA) macros to perform such tasks as automating repetitive modeling or post-processing tasks or embedding customized design routines. Following an open architecture paradigm, OpenSTAAD was built using ATL, COM, and COM+ standards as specified by Microsoft. This allows OpenSTAAD to be used in a macro application like Microsoft Excel or Autodesk AutoCAD. OpenSTAAD can also be used to link STAAD data to Web-based applications using ActiveX, HTML, and ASP.

OpenSTAAD allows engineers and other users to link in-house or third-party applications with STAAD.Pro. For example, a user might create a spreadsheet in Excel to analyze and design a circular base plate using support reactions from STAAD. With OpenSTAAD, a simple macro can be written in Excel or within the STAAD environment to retrieve the appropriate STAAD data and automatically link the results. If the STAAD file changes, so will the Excel sheet! With a built-in VBA editor, macros can be written inside STAAD using VBA to create new dialog boxes or menu items which run design codes or specific structural components (like certain connections) that automatically link to STAAD's familiar reporting tables. A cumbersome export/import link between two or three software is not required.

OS. Help and Documentation

OpenSTAAD Help Organization

The OpenSTAAD documentation is organized in the following manner:

- Getting Started
- Fundamentals
- Using Macros
- OpenSTAAD Application Objects
- Troubleshooting
- Examples
Printing Help and Documentation Files

Topics in the OpenSTAAD help system may be printed when they are opened in the Help Viewer application. To print a topic:

1. Select the topic you wish to print from the Table of Contents.
2. Click the Print tool.
   
   or
   
   Press <Ctrl+P>.

OS. Function List Organization

The list of functions is formatted in the following way:

- The name of each function appears in large, bold typeface at the top of the page.
- A general description follows which contains the purpose of the function, and provides general comments, constraints, and recommendations for using it.
- The Visual Basic (VB) syntax for the function appears next in bold typeface. All function syntax definitions consist of the function name and then the comma separated parameters, e.g.,
  functionname param1, param2, ... , paramn
  
The function and all its parameters are usually written on a single line, but for long lines of code, a line continuation character may be used to make the code easier to read. In VB, the line continuation character is a space followed by an underscore ( _ ).
- Following the function syntax definition is a description of each parameter required by the function. Note that the names of the parameters are just example names. If the parameter is a variable name for storing results returned by the function, or if you are passing a parameter required by the function as a variable, you can give it any legal variable name.
- If the function directly returns a value (often a boolean value indicating if the function was successful or not), a section on interpreting the return value is included.
- Then an example of the function is provided. In some cases, the example is just a short code snippet; in other cases a fully viable macro or program is given. All code provided will be able to work within the STAAD macro editor or an external VBA editor (e.g., Microsoft® Office Excel®).
- Finally, a list of related functions are listed, where applicable.

OS. Fundamentals of OpenSTAAD

OS. Programming Languages

OpenSTAAD supports Visual Basic for Applications (VBA), which is an implementation of Microsoft's Visual Basic language and an associated IDE.

Although OpenSTAAD supports all major programming languages used today capable of calling COM components, it is impractical to document the usage of each and every function in all of these languages. Most of the example programs or code snippets for each documented OpenSTAAD function are written in VBA.
for Excel or AutoCAD VBA. The reason is that with OpenSTAAD 2.0 and higher, STAAD.Pro is equipped with a functional VBA editor capable of writing macros only in VBA.

Documentation for VBA is beyond the scope of this manual and Bentley does not provide direct support on how to write VBA macros. There are, however, several useful and free sites on the Web to assist a beginner on writing macros in STAAD, Excel, AutoCAD, or any other VBA compliant software. It should be noted that the VBA language is the same from software to software. However, the functions, objects, and core libraries will obviously vary.

The sites recommended by Bentley are as follows:

- [https://stackoverflow.com](https://stackoverflow.com)
- [https://www.oreilly.com/](https://www.oreilly.com/)
- [http://www.wrox.com](http://www.wrox.com)

Also, it is worth noting that many programs such as MicroStation and Microsoft Office Excel have a useful Record Macro feature. You can run the recorder and then select any commands from the program’s ribbon tabs. The corresponding VBA syntax will automatically be generated.

For additional information on using macros and VBA in Excel or other Microsoft Office programs, Microsoft’s MSDN is a recommended starting point:


**OS. OpenSTAAD User Forum and Code Resource**

Bentley hosts an OpenSTAAD community on the company's Bentley Communities website. Here you can exchange macros with peers using the file upload, ask questions in the community forum, or share other information in the community blog and image gallery.

Bentley hosts an OpenSTAAD community on the company's Bentley Communities website. Here you can exchange macros with peers using the file upload, ask questions in the community forum, or share other information in the community blog and image gallery.

Join the Bentley OpenSTAAD group at Bentley Communities today.

**Caution:** Code or files uploaded to Bentley Communities are not subject to a screening process in advance. Any files or code which are found to result in malicious behavior will be removed. Bentley Systems accepts no responsibility for any mistake, error or misrepresentation in or as a result of the usage of software code obtained from the Bentley Communities site.

**OS. Application Program Interface (API)**

The OpenSTAAD library of functions is classified under the following general categories:

- STAAD File Input and Output (I/O)
- Structure Geometry
- Member Specifications
- Properties
- Loads
- Output Results
OS. Instantiating the OpenSTAAD Library for Use

The first thing necessary to access STAAD project data from within another application is to instantiate, or create an instance of OpenSTAAD within the other application. In Visual Basic for Applications (VBA), this may be done by creating an object variable and then assigning to it the OpenSTAAD object. The VBA `GetObject` function may be used for this.

The object which controls the STAAD.Pro environment is referred to as `StaadPro.OpenSTAAD`. This object must be created in order to get access to any of the internal graphical functions within STAAD.Pro (including the creating of menu items and dialog boxes) as well as access to STAAD's viewing, geometry modeling, results grid, and post-processing functions. The following VBA function can be used to instantiate or create this object:

```vba
Set MyObject = GetObject("filepath", "objectclass")
```

Where:
- **MyObject** the Object name declared in a previous `Dim` statement.
- **filepath** the optional string providing the full file path and name of the file containing objects to retrieve.
  In the case of OpenSTAAD, this can be omitted, along with the trailing comma.
- **objectclass** the string representing the class of the object. In the case of OpenSTAAD, this is always `Staadpro.OpenSTAAD`.

Thus:
```vba
Set objName = GetObject("StaadPro.OpenSTAAD")
```

At the conclusion of your OpenSTAAD application, the OpenSTAAD object(s) should be terminated, to unlock the handles within the application to the OpenSTAAD functions, and to free system resources.

**Example**

```vba
Sub How2Begin()
    'Create a variable to hold your OpenSTAAD object(s).
    Dim objOpenSTAAD As Object
    'Launch OpenSTAAD Object
    Set objOpenSTAAD = GetObject(, "StaadPro.OpenSTAAD")
    'At the end of your application, remember to terminate the OpenSTAAD objects.
```

Set objOpenSTAAD = Nothing
End Sub

OS. Function Return Value

Most OpenSTAAD functions return a value to either:

- indicate the success or failure of the function (a Boolean result), or
- results value of the function (a numeric value result).

If a function returns a Boolean result and that return value for an OpenSTAAD function is equal to 0 (zero), it means that OpenSTAAD was unable to execute the function. If you see this result, check that you have passed all the required parameters to the function. Make sure that all parameters being passed are valid. A return value of 1 (one) indicates that OpenSTAAD successfully executed the function. Unless specified otherwise, results returned by a function are stored in variable names passed to it for the purpose.

A few of the OpenSTAAD Application functions return the results as the return value of the function. In those cases, the above comments regarding the function return value do not apply.

If a function returns the value of the function,

```vbnet
Dim returnValue
returnValue = objOpenSTAAD.BooleanReturn (param1, param2, … ,paramn)
```

Tip: See “Functions and Subroutines” (on page 5050) for information on parenthesis required in VB Syntax.

OS. STAAD Nomenclature

In STAAD documentation and in the program menus and dialog boxes, the terms “member” and “beam” are used interchangeably. Use of the term "beam" should not be taken to imply that the member cannot take an axial load. Similarly, the terms "joint" and "node" are used interchangeably in STAAD. Both terms refer to the connections between elements.

Connections are also referred to as incidences. The terms “member incidences,” “plate incidences,” and “solid incidences” refer to the nodes that connect these elements to other elements and supports.
OS. OpenSTAAD Compatibility with STAAD.Pro

For optimum performance, OpenSTAAD should be run in STAAD.Pro 2002 or higher. The object which controls the STAAD.Pro graphical interface (StaadPro.OpenSTAAD - dialogs, modeling tools, etc.) is only available in STAAD.Pro 2003 or higher.

As software development proceeds, it is sometimes necessary to modify STAAD's database to accommodate new features or to maximize efficiency and increase processing speed. STAAD.Pro 2002 is the first STAAD release to include OpenSTAAD. All subsequent releases of OpenSTAAD will be backward-compatible with STAAD.Pro 2002.

It may be possible to run at least some of OpenSTAAD's functions in earlier releases of STAAD.Pro or STAAD-III, but the results may be unpredictable, since OpenSTAAD did not exist at that time, and therefore compatibility with OpenSTAAD was not considered during the development of these earlier STAAD versions. Bentley Systems, Inc. does not provide any technical support for OpenSTAAD running in STAAD versions prior to STAAD.Pro 2002.

In terms of STAAD's input file, STAAD.Pro is backward-compatible with all previous STAAD releases. To use OpenSTAAD on a project that was created using an earlier version of STAAD.Pro or STAAD-III, you can open your old input file in STAAD.Pro 2002 and run the analysis. This action will create a new database in a format fully compatible with OpenSTAAD.

OpenSTAAD is compatible with any version of Microsoft® Office Excel® that supports VBA macros. OpenSTAAD is also compatible with Autodesk AutoCAD® 2000 (or higher) VBA.

With OpenSTAAD 2.0 and higher (available with STAAD.Pro 2003 and higher), a VBA editor is embedded within the STAAD.Pro environment. The version of VBA which the editor supports is not 100% VBA compatible. There are a few functions which can be supported in the Microsoft version of VBA which are not supported in the VBA editor/compiler which comes with STAAD.Pro. Every attempt is being made to support these functions. Currently, the functions which are not supported are not yet documented in this manual.

OS. Visual Basic Conventions

Comments

In Visual Basic or Visual Basic for Applications, an apostrophe (’) is used to denote a comment. Anything to the right of the apostrophe will be ignored by the program.

Tip: Throughout this documentation, these are displayed in the color green to make comments visually distinct from the remaining code.

Declaring Arrays

VB/VBA is flexible in the way it allows arrays to be declared. Most examples involving arrays in this reference manual will conform to the C++ zero indexing convention. In an array of 6 values, the positions in the array are referred to as 0-5. Therefore an array of 6 values would be declared as follows:

Dim pdArray(5) As Double

or

Dim pdArray(0 To 5) As Double
In VB, a six value array can also be declared as:

```vbnet
Dim pdArray(1 To 6) As Double
```

In doing so, however, we might find that our loops and other statements used to access the various positions in the array might not work correctly in C++.

**Line Continuation Character**

A long coding statement can be written on more than one line, to make the code easier to read. The VB line continuation character consists of a space followed by an underscore at the end of the line. The line of code beneath the continuation character will be handled as though it was written on the same line as the line continuation character.

**Functions and Subroutines**

Though functions and subroutines perform similar actions in any language, it is important to understand a key difference in order to make writing OpenSTAAD macros easier. A function can return a value back to the function or routine that called it. A subroutine, on the other hand, does not directly return a value. Instead, a subroutine must make use of the ByRef methodology internally. OpenSTAAD has been written so that you can use many of its functions to return more than one value via a single function. While this is a very powerful feature, it can lead to some confusion given how VB syntax uses parenthesis. In VB syntax, a subroutine does not use parenthesis to enclose any values which are passed to it. VB reserves their use for functions only.

Thus, in OpenSTAAD the subroutines which are used to return multiple values from the model input, analysis results, or design results are required to use the subroutine syntax.

**Note:** As this is a merely a result of VB syntax and not that these OpenSTAAD functions behave differently, the documentation simply refers them all as "functions."

**Tip:** Often, though, the data passed in such a function is in the form of an array, which do use parenthesis (without a space) to indicate their dimensions.

```
Example

objOpenSTAAD.GetNodeDisplacements nNodeNo, nLC, pdDisps(0)
' The line of code above may also be written as the line shown below:

objOpenSTAAD.GetNodeDisplacements _
nNodeNo, nLC, pdDisps(0)
```

**OS. Using Macros in STAAD.Pro**

This section describes how to use the STAAD.Pro Script Editor to create a new macro or load existing macros in STAAD.Pro. It also describes how to run the macros you have created in the STAAD.Pro Script Editor by adding a simple menu item.

**OS. To start a new macro project**

To create a new macro project and open it in the STAAD.Pro Script Editor, use the following procedure.
1. On the **Utilities** ribbon tab, select the **Macro** tool in the **Developer** group.

   - The **Macro** dialog opens.

2. Click **Create New**.
   - The **New Macro File Name** dialog opens.

3. Type a **File name** and optional **Description of Macro**.
   - For this tutorial, type **Frame** in the **File name** field and then type **Create a 2D frame with supports** in the **Description of Macro** field.

4. Click **New**.
   - The STAAD.Pro Script Editor window opens. An empty script is loaded with for your file.

OS. To import an existing macro

1. On the **Utilities** ribbon tab, select the **Macro** tool in the **Developer** group.

   - The **Macro** dialog opens.

2. Click **Import**.
   - The **Add an Existing Macro** dialog opens. The controls in this dialog are analogous to those in the **Select New Macro File Name** dialog, though some are inactive.

3. Navigate to and select the existing macro file.

4. Click **Open**.

   - The macro name is added to the list of names in the Macro dialog.

**Related Links**

- **EX. OpenSTAAD Example Files** (on page 4311)

OS. To run a linked macro

1. On the **Utilities** ribbon tab, select the **User Tools** tool in the **User Tools** group.

   - A drop-down menu displays all of the linked macros.

2. Select the macro you want to run from this list.

OS. Displaying OpenSTAAD Functions in the Objects Browser

The objects browser is very useful to see an overview of functions while writing OpenSTAAD macros.
1. Open the SAX Basic editor.

   **Tip:** Start a new macro or edit an existing one.

2. Right click anywhere in the window and select **Edit > References** from the pop-up menu.

   The **References** dialog opens.

   ![References Dialog](image)

   3. Check the option for **OpenSTAADUI (1.0)** in the Available References list and click **OK**.

   Now, when the **Browse Object** tool is selected, you will see OpenSTAAD functions included in the Library list.
OS. STAAD.Pro Script Editor window

This application is used to write macros for use with STAAD.Pro. It is a small yet powerful programming application with tools for helping you create and debug macros.

Tip: The STAAD.Pro Script Editor is built on the WinWrap® Basic Editor platform. Select the Help tool in the top-right corner of the application to get help on both the WinWrap application as well as the WinWrap Basic language, which supports Visual Basic .NET™.
## OS. Home ribbon tab

Table 663: Clipboard group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Paste</td>
<td>Pastes the clipboard contents (text only) at the cursor position.</td>
<td>&lt;Ctrl+V&gt;</td>
</tr>
<tr>
<td>Cut</td>
<td>Copies the selected contents to the clipboard and deletes the original.</td>
<td>&lt;Ctrl+X&gt;</td>
</tr>
<tr>
<td>Copy</td>
<td>Copies the selected contents to the clipboard.</td>
<td>&lt;Ctrl+C&gt;</td>
</tr>
</tbody>
</table>

Table 664: Edit group

<table>
<thead>
<tr>
<th>Tool Name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select All</td>
<td>Selects the entire contents of the programming area.</td>
<td>&lt;Ctrl+A&gt;</td>
</tr>
<tr>
<td>Undo</td>
<td>Undoes the last action in the editor.</td>
<td>&lt;Ctrl+Z&gt;</td>
</tr>
<tr>
<td>Redo</td>
<td>Reverses the last undo action.</td>
<td>&lt;Ctrl+Y&gt;</td>
</tr>
<tr>
<td>Indent</td>
<td>Indents the selected line or lines.</td>
<td></td>
</tr>
<tr>
<td>Outdent</td>
<td>Outdents the selected line or lines.</td>
<td></td>
</tr>
<tr>
<td>Comment</td>
<td>Changes the current line (or lines, if a selection spans multiple lines) to a comment. These lines are then ignored by the compiler.</td>
<td></td>
</tr>
<tr>
<td>Uncomment</td>
<td>Changes a commented line or lines back to commands to be interpreted by the compiler.</td>
<td></td>
</tr>
</tbody>
</table>
### Table 665: Search group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Find</td>
<td>Opens the Find dialog, which is used to find strings that match the dialog input.</td>
<td>&lt;Ctrl+F&gt;</td>
</tr>
<tr>
<td>Replace</td>
<td>Opens the Replace dialog, which is used to replace any matched strings with a different string.</td>
<td>&lt;Ctrl+H&gt;</td>
</tr>
<tr>
<td>Goto</td>
<td>Opens the Go To Line dialog, which is used to move the cursor to a specified line number.</td>
<td>&lt;Ctrl+G&gt;</td>
</tr>
</tbody>
</table>

### Table 666: Execute group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run</td>
<td>Runs the current macro.</td>
</tr>
<tr>
<td>Pause</td>
<td>Pauses running the current macro. Click again to resume.</td>
</tr>
<tr>
<td>Stop</td>
<td>Stops running the current macro.</td>
</tr>
</tbody>
</table>

### Table 667: Debug group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Toggle Design Mode</td>
<td></td>
</tr>
<tr>
<td>Toggle Breaks</td>
<td></td>
</tr>
<tr>
<td>Clear Breaks</td>
<td></td>
</tr>
<tr>
<td>Step Into</td>
<td></td>
</tr>
<tr>
<td>Step Out</td>
<td></td>
</tr>
<tr>
<td>Step Over</td>
<td></td>
</tr>
<tr>
<td>Current Statement</td>
<td></td>
</tr>
</tbody>
</table>
### Table 668: Design group

<table>
<thead>
<tr>
<th>Tool name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>User Dialog</td>
<td>Opens the UserDialog Editor, which is used to create end-user dialogs for your macro.</td>
</tr>
<tr>
<td>Add References</td>
<td>Opens the References dialog, which is used to include references for APIs, including OpenSTAAD.</td>
</tr>
<tr>
<td>Browse Object</td>
<td></td>
</tr>
<tr>
<td>View Objects</td>
<td></td>
</tr>
</tbody>
</table>

**OS. UserDialog Editor**

Used to create end-user dialogs for your macro. This will generate the `Begin Dialog…End Dialog` block in the macro code.

The editing tools are located along the top toolbar.

The user interface items are on tool pallet on the left side. Click any tool to active that item for layout. Then click in the dialog view to place that tool.

**OS. References dialog**

Used to include references for APIs, including OpenSTAAD.

**OS. OpenSTAAD API Documentation**

Programmer reference documentation for the OpenSTAAD API is installed separately at C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\Help\OSAPP\index.html (typical location).

1. Select the **File** ribbon tab.  
   The backstage view opens.

2. Select **Help > OpenSTAAD Help**.  
   The OpenSTAAD Documentation opens in your default web browser.

**OS. Troubleshooting**

The following is a list of error messages or warnings which may be displayed and methods used to address them.

**OS. Method Object Failed**

Error message:
Method 'OpenSTAADFuntionName' of object IOutput failed.

where OpenSTAADFuntionName is the name of an OpenSTAAD function. For example:
Method 'GetMemberBetaAngle' of object IOutput failed.

**Solution**

Check to be sure that you are passing all parameters required by the function. If a parameter is missing, or if the parameter being passed is not valid, the Method ...of object IOutput failed message may appear. For example, the message may appear if a member number is passed to the function, but that member number does not exist in the currently open STAAD file.

You may also receive similar messages if your version of STAAD.Pro is not compatible with OpenSTAAD. OpenSTAAD is compatible with STAAD.Pro 2001 Build 1006.US.REL and higher. You will need to reanalyze your model in that build. This is because we switched the way the data is recorded in Build 1006 to save more space. Build 1006 US.REL was released in November 2001.

**Note:** UK Builds prior to STAAD.Pro 2002 are not compatible with OpenSTAAD.

**OS. Function is not retrieving correct values**

Check to be sure that you have saved the STAAD file after making changes to the input before you run any OpenSTAAD functions that retrieve input information. If you are running STAAD functions that retrieve analysis results, check to be sure that you have saved the STAAD file and re-run the analysis after making any changes to the input.

**OS. Type Mismatch**

Error message:
Type mismatch

Check to make sure that you have declared all variables using the DIM statement at the beginning of your program or macro, e.g.:

```
Dim pnIsReleased As Integer
Dim pdSpringStiffnesses(0 To 5) As Double
Dim pdPartialMomRelFactors(0 To 2) As Double
```

Confirm that when you pass array variables to the function, you have specified the starting position in the array for the function to use in filling up the array. For example:

```
objOpenSTAAD.GetFullMemberReleaseInfoAtStar 3, pnIsReleased, _
pdSpringStiffnesses(0), pdPartialMomRelFactors(0)
```

**OS. Property or Method Not Supported**

Error message:
Object doesn't support this property or method

Check to make sure you have typed the function name correctly.
OS. ActiveX Component in Microsoft® Excel

Error message:
ActiveX component can't create object
when attempting to run an OpenSTAAD macro in Microsoft Excel.

Are macros enabled?
When you open the Excel file, is it asking you to Enable Macros?
If so, make sure you click Yes. If not, you need to change the security settings in Excel. Select Tools > Macro > Security... from the Excel menu. Make sure the Security Level is set to Medium. Close Excel down (completely, not just the file) and reopen the file again. This time it should ask you if you want your Macros enabled or disabled. Choose Enabled.

OpenSTAAD DLL not registered
When you run your macro, if it still gives you the error ("ActiveX component can’t create object"), it might be because the OpenSTAAD library was not registered properly when the program was installed.

Register the DLL
1. In Windows Explorer, go to the following location:
   \C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\Plugins\Struclink\.
   
   Note: The path to the file OpenStaad.Registration.exe may be different on your computer if you did not use the default installation path provided by the STAAD.Pro installer when you installed your software – if so, go to folder corresponding to the actual location on your computer.

   3. Right-click on the file OpenStaad.Registration.exe and select Run as Administrator from the pop-up menu.
   4. Click Register OpenSTAADUI to register the DLL.
   A message dialog opens indicating that the OpenSTAADUI type library is registered. If the registration did not succeed, please contact our technical support staff for further instructions.
   5. Click OK.

   6. Try opening and running the Microsoft Excel beam example file provided with your STAAD.Pro software (C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\OpenSTAAD\Rectangle-beam.xls).
   7. Close all the STAAD.Pro instances and open only the relevant STAAD.Pro model.
   8. Run the macro.

OS. User Type Not Defined

The message may be appearing because the OpenSTAAD library references have not been included. The VBA compiler therefore does not know which functions are associated with the OpenSTAAD object.

There are two ways to eliminate this error message:
1. Declare the OpenSTAAD object *As Object*, instead of *As Output*, e.g.
   
   ```vba
   Dim objOpenSTAAD As Object
   ```

2. Include the OpenSTAAD library reference in your VBA editor. This second option has the added benefit that whenever you do it, the compiler will now recognize the OpenSTAAD object and will pop up a list of functions whenever you refer to the object in your VBA editor.

   To include the OpenSTAAD library reference, select *Tools > References* in your VBA editor. A dialog titled *References – Normal* opens. You will see a scroll box inside the dialog box labeled *Available References*. Scroll down through the list of references until you find and entry labeled *OpenSTAAD 1.0 Type Library*. Toggle on the corresponding check box, then click *OK*.

   Now re-run your macro to see if the problem with the *User Defined type not defined* error message has been solved.

**OS. Files Not Compatible**

**Error message:**

One or more results files are not compatible with the current model and cannot be loaded...

You need to have a successful analysis before you can run any OpenSTAAD macro, including such simple macros as one that only asks for the number of nodes in the model. The program is not smart enough to know which commands will work and which will not when no results are present. It first checks to see if valid results are available. If they are not, it displays the error message and terminates.

You can use the OpenSTAAD command *AreResultsAvailable* in your macro to check to see if results are available. In some cases, this function also will result in the error message *One or more results files are not compatible with the current model and cannot be loaded*.... That means that even though the STAAD results file with an ANL extension has been created, the file contains only error messages and no meaningful results.

You can also try inserting a *FINISH* command in your input file preceding the location in the input that is causing the analysis errors. Depending on whether the errors take place before any analysis is performed, you might get rid of the original error message, but in its place you may see the message, *No Results Available*. However, the best way to eliminate this problem is to modify your input file so that you obtain a successful analysis before you try to run your OpenSTAAD macro.

Under certain conditions, you might get this *...results files are not compatible...* error message, even though the analysis runs successfully. OpenSTAAD currently will not work with the Moving Load Generator. The reason is that the results from the Moving Load Generator are not kept in the same database as the other STAAD results. Therefore, when OpenSTAAD tries to read the Output File, it is not able to find those missing load results, so it displays an error message and stops processing the macro. This same situation is in effect for any other loads that you are unable to see until you perform the analysis, like UBC loads, for example. If the loads are defined in the input file, OpenSTAAD will work fine, but not with loads that are only generated when the analysis is run, because these results are not kept in a location where OpenSTAAD can find them.

If you remove your moving load generation command from your input file, and run the analysis, you should then be able to run your OpenSTAAD macro.

**OS. Macro Tutorial**

This tutorial will take you step-by-step through the process of creating a practical macro used to generate a parametric 2D frame with supports.
The assumptions behind this parametric frame are:

- 2D frame lies in the XY plane at Z = 0 (with first support node at the origin)
- All bays are of equal width and height
- The user specifies the units of length before running the macro
- All supports have the same type and all nodes at Y = 0 are supports

Most of these assumptions could be either validated for altered with some changes or additions to the macro presented in this tutorial.

OS. To start a new macro project

To create a new macro project for this tutorial and to then open it in the STAAD.Pro Script Editor, use the following procedure.

The macro used in this tutorial is intended to run in an empty file. Therefore, you should start with an new STAAD input file in the Analytical Modeling workflow.

1. On the Utilities ribbon tab, select the Macro tool in the Developer group.
The Macro dialog opens.

2. Click Create New.
   The New Macro File Name dialog opens.

3. Type a File name and optional Description of Macro.
   For this tutorial, type Frame in the File name field and then type Create a 2D frame with supports in the Description of Macro field.

4. Click New.
   The STAAD.Pro Script Editor window opens. An empty script is loaded with for your file.

5. Select the line ‘TODO: Add your code here (i.e., line 4) and press <Delete>.

OS. Creating the User Dialog

The form is the dialog which a user will use to specify the frame parameters. This form will used text entry fields for the frame size and then a pair of options to allow the user to select the support type.

OS. To create the dialog form

Your cursor should be located where the dialog code will be inserted. If you deleted the comment line in the previous procedure, then your cursor is in the correct location (i.e., line 4).

1. Select the User Dialog tool in the Design group.

The UserDialog Editor opens.

2. Either:
   Click the Edit Item Properties tool
   or
   Right-click anywhere in the form layout
   The Edit User Dialog Properties dialog opens.

3. Type 600 in the Width field and then 200 in the Height field.
   These values are somewhat arbitrary, but allow for a sufficient amount of space to include the dialog controls.

4. Type 2D Frame Model in the Caption field.
5. Click Close.

6. Select the Set Grid tool.

   The Grid Settings dialog opens.

   7. Type 5 for both the Horizontal and Vertical grid values.

8. Click OK.
OS. To create the text fields and labels

1. Select the **Add Text** tool.
2. Click somewhere near the top, left corner of the form.

3. Repeat steps 2 and 3 to add four additional text labels along the left side of the dialog. This does not have to be precise. The labels and fields will be aligned by coordinates in a later step.

4. Select the **Add TextBox** tool.
5. Click a point to the right of the first label to add a data field in the form.
6. Repeat steps 4 and 5 to add three additional text boxes.
7. Edit the location and caption for the first label:
   a. Right-click on the first label.
      The **Edit Text Properties** dialog opens.

      ![Edit Text Properties dialog]

   b. Type the following values to define the label layout:
      - Left = 20
      - Top = 20
      - Width = 190
      - Height = 15 (default if you placed with a click)
   c. Type No. of Horizontal Bays: in the **Caption** field.
      Leave the remaining fields as their defaults.
   d. Click >>.
The properties for the next control added are displayed. This allows you to quickly make changes to the controls without the need to close and reopen this dialog.

8. Repeat step 7 to update the layout and caption for the remaining text labels as indicated below.

<table>
<thead>
<tr>
<th>Label Name</th>
<th>Rectangle (Left, Top, Width, Height)</th>
<th>Label Text</th>
</tr>
</thead>
<tbody>
<tr>
<td>Text2</td>
<td>20,45,190,15</td>
<td>No. of Vertical Bays:</td>
</tr>
<tr>
<td>Text3</td>
<td>20,70,190,15</td>
<td>Vertical Distance:</td>
</tr>
<tr>
<td>Text4</td>
<td>20,95,190,15</td>
<td>Horizontal Distance:</td>
</tr>
<tr>
<td>Text5</td>
<td>20,130,190,15</td>
<td>Support Type:</td>
</tr>
</tbody>
</table>

When you click >> on Text5, the Edit TextBox Properties dialog opens for the first text box control.

9. Edit the location and name for the text box:
   a. Type the following values to define the text box:
      
      Left = 220
      Top = 20
      Width = 130
      Height = 15
   b. Type clmn in the Field field.
      This is the object name which the macro will use for this text box.
   c. Click >>.
      The properties for the next control added are displayed.

10. Repeat step 9 to update the layout and field name for the remaining text boxes as indicated below.

<table>
<thead>
<tr>
<th>Text Box Name</th>
<th>Rectangle (Left, Top, Width, Height)</th>
<th>Field Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>TextBox2</td>
<td>220,40,130,15</td>
<td>row</td>
</tr>
</tbody>
</table>
11. Click Close.

OS. To create support options

Since the macro in this tutorial limits the support options to only fixed or pinned, using a pair of radio buttons to allow the user to select the support type is a good design choice. However, in a situation where three or more options were available, a drop-down list control might be a better choice.

1. Select the Add OptionButton tool.
2. Click a point to the right of the Support Type label (i.e., text5).
   The option control is placed.
3. Repeat steps 1 and 2 to add a second option to the right of the first.
4. Right-click on the first option control.
   The Edit OptionButton Properties dialog opens.
5. Type the following values to define the location:

   Left = 220
   Top = 130
   Width = 90
   Height = 15

6. Type Fixed in the Caption field.
7. Type sprt in the OptionGroup combo box.
8. Click >>.
   The properties for the second option control are displayed.
9. Repeat steps 5 through 7 except type 370 in the Left field and Pinned in the Caption field.

   Tip: Since the option group was previously defined, you can now selected from the drop-down list in the combo box rather than re-type its name.

10. Click Close.
    The Edit OptionButton Properties dialog closes.
To add the dialog buttons

The last step in defining the dialog is to add the OK and Cancel buttons.

1. Click the **Add OK Button** tool.
2. Click a point near the bottom, right corner of the form. The OK button is placed.
3. Click the **Add Cancel Button** tool.
4. Click a point to the right of the OK button.

*Note:* This is the recommended button order according to Microsoft's design guidelines.

The Cancel button is placed.

5. (Optional) Either:
   - right-click on either button to precisely edit the location
   - drag the buttons to align them to the grid

6. Select the **Save and Exit** tool.
   - The **UserDialog Editor** closes.

The dialog definition code is added to your macro:

```
Begin Dialog UserDialog 600,200,"2D Frame Model" %GRID:5,5,1,1
Text 20,20,190,15,"No. of Horizontal Bays:",.Text1
Text 20,45,190,15,"No. of Vertical Bays",.Text2
```

**OS. Macro Tutorial**
Tip: Now is a good time to save the work done to this point.

OS. To dimension the variables and add initial values in the dialog

You must dimension all the values you will use in the program. Further, it's a good idea to populate the dialog with initial values for the user.

1. Select the automatically generated line of code `Dialog  dlg` (i.e., line 23) and delete the line contents.
2. With your cursor after the line of code `Dim  dlg  as  UserDialog`, type the following to dimension the variables in use:

   ```vba
   Dim dlgResult As Integer
   Dim crdx As Double
   Dim crdy As Double
   Dim crdz As Double
   Dim n1 As Long
   Dim n2 As Long
   Dim ii As Long
   Dim s1 As Long
   ```

   **Note:** As you begin to type, the IntelliSense will provide options for the variable types. You can make use of the IntelliSense to help quickly complete lines and reduce typos in your source code.

3. Type a single quote mark followed by `Initialization`. 
The single quote mark is used to denote a comment. It is best practice to add clear comments to document your source code.

4. Type the following values to populate the dialog input fields:

   `dlg.clmn = "3"
   dlg.row = "5"
   dlg.ht = "3"
   dlg.wdth = "5"

5. Type the following commands to open the dialog upon starting the macro.

   You'll also add a comment and a debugging command which is helpful for resolving any issues with your macro.

   `DlgResult = Dialog(dlg)
   Debug.Clear`

6. Type the following command to capture the result of the user action in the dialog:

   `If dlgResult = -1 Then 'OK button pressed
   Debug.Print "OK button pressed"
   ElseIf dlgResult = 0 Then 'Cancel button pressed
   Debug.Print "Cancel button pressed"
   End If`

   **Note:** The single quote mark after the world Then also indicates a comment. Comments like this can follow the programming instruction on the same line.

   The EndIf statement is automatically added by the editor to close the If statement if you press <Enter>.

You have now created the general set of instructions to the program what to do with the user action in the form.

You can test the macro now to see the dialog populated with the initialized values. Select the Run tool in the Execute group. The macro dialog opens. Click either OK or Cancel to close it. Neither perform any action at this point other than to log a debug message.

The next step is to have the script then perform the desired actions when the users clicks OK.

**Tip:** Now is a good time to save the work done to this point.

**OS. To get the user values**

1. Place your cursor just after the Debug.Print "OK button pressed" line and then press <Enter> several times to create additional space between this and the following line beginning with ElseIf.

2. Type the following commands to populate the variables with the values the user types into the dialog:

   `'Get the values
   clmn = Abs( CDb1(dlg.clmn) )
   row = Abs( CDb1(dlg.row) )
   ht = Abs( CDb1(dlg.ht) )
   wdth = Abs( CDb1(dlg.wdth) )
   sprt = CStr(dlg.sprt)`
Note: The last captures the support array position as a string whereas the others all capture absolute value of
the user input for the respective data fields.

3. Type the following debug statements to provide back the captured values:

   Debug.Print "No. of Horizontal Bays = ";clmn
   Debug.Print "No. of Vertical Bays = ";row
   Debug.Print "Vertical Distance = ";ht
   Debug.Print "Horizontal Distance = ";wdth
   Debug.Print "Support Type = ";sprt

4. Type the following command to set the initial values for the coordinate variables:

   crdx = 0
   crdy = 0
   crdz = 0

OS. To initialize OpenSTAAD and calculate the node coordinates

1. Type the following command to initialize OpenSTAAD:

   Set staad = GetObject(,"staadpro.openstaad")

   This command opens the connection to STAAD.Pro and the model currently open in the program.

   Note: You can initialize OpenSTAAD to any variable. Here, staad is used.

2. Type the following two For loops to calculate the nodal coordinates for each node in the frame.

   Note: There are two nested loops used because this is a 2D frame. The Z coordinate was previously set to
   zero and will not change.

   'Nodes
   For j = 2 To (row + 2)
       For i = 1 To (clmn + 1)
           crdx = (i - 1) * wdth
           staad.Geometry.AddNode crdx, crdy, crdz
       Next
       crdy = (j - 1) * ht
   Next

   Note: The Next command to close the For loop is added automatically once you press <Enter> after typing
   the For command line.

3. Type the following If ElseIf statement to get the desired support type.

   The support is created and the returned support reference number is stored to the variable s1.

   A “fall back” possibility is also added in case neither of the intended support types is somehow specified, then
   an error message is presented to the user.

   'Supports
   If sprt = "0" Then
       s1 = staad.Support.CreateSupportFixed()
   ElseIf sprt = "1" Then
       s1 = staad.Support.CreateSupportPinned()
   Else
       MsgBox("Select Proper Support Type",vbOkOnly,"Error")
       Exit Sub
   End If
**Note:** The OpenSTAAD command `staad.Support.CreateSupportFixed` can also be used to simply create a new support without using the return value. But by adding the parenthesis at the end, the script can take the returned support reference number which allows you to store that in the variable.

4. Type the following line to add a debugging statement for the returned support reference number:

   ```vbscript
   Debug.Print "Support return value = "; s1
   ```

5. Type the following type to assign the support types to the bottom nodes in the frame:

   ```vbscript
   For i1 = 1 To (clmn + 1)
   staad.Support.AssignSupportToNode i1, s1
   Next
   ```

**Tip:** Now is a good time to save the work done to this point.

**OS. To generate the frame members**

1. Type the following commands to add the columns:

   ```vbscript
   'Columns
   n1 = 1
   n2 = (n1 + clmn + 1)
   For k = 1 To (clmn + 1)*row
       staad.Geometry.AddBeam n1, n2
       n1 = n1 + 1
       n2 = n2 + 1
   Next
   ```

2. Type the following commands to add the beams:

   ```vbscript
   'Beams
   n1 = 1
   For k1 = 1 To row
       n1 = k1 * (clmn + 1)+1
       n2 = n1 + 1
       For k2 = 1 To clmn
           staad.Geometry.AddBeam n1, n2
           n1 = n1 + 1
           n2 = n2 + 1
       Next
   Next
   ```

3. Save your macro by either:

   - click the **File** ribbon tab and then select the **Save** tool in the backstage view
   - or
   - click the **Save** tool in the quick access toolbar
   - or
   - press `<Ctrl+S>`

Your frame macro is now complete. You are ready to test the macro to check your code.
OS. To test your macro

You can add break points through your code to assist in debugging. Practice adding one so you can read the debug print output you added.

If the program contains errors, the line where the program stops will be marked in red. Stop the program and then review your input to debug.

1. Click in the dark gray margin of the code editor just to the left of line number 57 (or the line containing cdrx = 0).
   A red dot appears to indicate this is a break point. The code will pause the execution at this point when executed.
2. Click the Run tool in the Execute group.

   The user dialog opens for input.
3. Change one or more values from their defaults.
4. Click OK.
   The code runs to the break point, which is now highlighted in yellow. The Immediate tab now shows the debug print statements. These should reflect any changed input you provided.

5. Click the Run tool in the Execute group again to continue to the next break point or to the end of the code.
6. Save your changes.
7. Select the File ribbon tab and then select Exit in the backstage view.

Your new macro is shown in the Macro dialog list.

**Note:** Testing the macro still runs within STAAD.Pro through OpenSTAAD, so you will have a frame generated if you tested it. You will need to start with a new, empty file to run the macro correctly.
OS. To add the macro to the list of user tools

Once you have completed testing of your macro, you can add it to the list of User Tools in the STAAD.Pro application. This allows you to run it with one click.

1. On the Utilities ribbon tab, select the Configure tool in the User Tools group.

   The Customize User Defined Tools dialog opens.

2. Click New.

3. Type 2D Frame Model in the name of the command.

4. Click [...] next to the Command field.

5. Navigate to and select your Frame.vbs macro file.

6. Click Open.

7. Click OK.

   Note: There are no parameters of external files used by this macro, so those fields are left empty.

Related Links
• AD.2006.6.12 User Tools (on page 336)

OS. To run the Frame macro

1. Create a new analytical modeling input file.

2. (Optional) Change the units of length to a convenient value.

3. On the Utilities ribbon tab, select the User Tools > 2D Frame Model tool in the User Tools group.

   The 2D Frame Model dialog opens.

4. Type the number of bays in the horizontal and vertical directions.

5. Type the spacing of the bays in the horizontal and vertical directions (X and Y, respectively).

6. Select the Support Type.

7. Click OK.

   The frame with supports is generated.

OS. Frame.vbs Macro

This is the full contents of the Frame.vbs file. You can copy this by clicking Copy to Clipboard below and paste into the STAAD.Pro Macro Editor window.

`/*
*--------------------------------------------------------------------------------------
OpenSTAAD
OS. Macro Tutorial
STAAD.Pro 5075 User Manual
*/
Sub Main()
'DESCRIPTION:Create a 2D frame with supports

Begin Dialog UserDialog 600,200,"2D Frame Model" ' %GRID:5,5,1,1
Text 20,20,190,15,"No. of Horizontal Bays":,.Text1
Text 20,45,190,15,"No. of Vertical Bays":,.Text2
Text 20,70,190,15,"Vertical Distance":,.Text3
Text 20,95,190,15,"Horizontal Distance":,.Text4
Text 20,130,190,15,"Support Type":,.Text5
TextBox 220,20,130,15,.clmn
TextBox 220,40,130,15,.row
TextBox 220,70,130,15,.ht
TextBox 220,95,130,15,.wdth
OptionGroup .sprt
   OptionButton 220,130,90,15,"Fixed",.OptionButton1
   OptionButton 370,130,90,15,"Pinned",.OptionButton2
OKButton 360,165,90,20
CancelButton 490,165,90,20
End Dialog
Dim dlg As UserDialog
Dim dlgResult As Integer
Dim crdx As Double
Dim crdy As Double
Dim crdz As Double
Dim n1 As Long
Dim n2 As Long
Dim I1 As Long
Dim s1 As Long

'Initialization
dlg.clmn = "3"
dlg.row = "5"
dlg.ht = "3"
dlg.wdth = "5"

'Popup the dialog
dlgResult = Dialog(dlg)
Debug.Clear
If dlgResult = -1 Then 'OK button pressed
    Debug.Print "OK button pressed"
    clmn = Abs( CDbI(dlg.clmn) )
    row = Abs( CDbI(dlg.row) )
    ht = Abs( CDbI(dlg.ht) )
    wdth = Abs( CDbI(dlg.wdth) )
    sprt = CStr(dlg.sprt)
    Debug.Print "No. of Horizontal Bays = ";clmn
    Debug.Print "No. of Vertical Bays = ";row
Debug.Print "Vertical Distance = ";ht
Debug.Print "Horizontal Distance = ";wdth
Debug.Print "Support Type = ";sprt

crdx = 0
crdy = 0
crdz = 0

Set staad = GetObject(,"staadpro.openstaad")

'Nodes
For j = 2 To (row + 2)
    For i = 1 To (clmn + 1)
        crdx = (i - 1) * wdth
        staad.Geometry.AddNode crdx, crdy, crdz
    Next
    crdy = (j - 1) * ht
Next

'Supports
If sprt = "0" Then
    s1 = staad.Support.CreateSupportFixed()
ElseIf sprt = "1" Then
    s1 = staad.Support.CreateSupportPinned()
Else
    MsgBox("Select Proper Support Type",vbOkOnly,"Error")
    Exit Sub
End If
Debug.Print "Support return value = ";s1
For i1 = 1 To (clmn + 1)
    staad.Support.AssignSupportToNode i1,s1
Next

'Columns
n1 = 1
n2 = (n1 + clmn +1)
For k = 1 To (clmn + 1)*row
    staad.Geometry.AddBeam n1, n2
    n1 = n1 + 1
    n2 = n2 + 1
Next

'Beams
n1 = 1
For k1 = 1 To row
    n1 = k1 * (clmn + 1)+1
    n2 = n1 + 1
    For k2 = 1 To clmn
        staad.Geometry.AddBeam n1, n2
        n1 = n1 + 1
        n2 = n2 + 1
    Next
Next

ElseIf dlgResult = 0 Then 'Cancel button pressed
    Debug.Print "Cancel button pressed"
End If
Alternate Control

If a user is asked to choose among three or more options, often a drop-down list is a better choice of control for the user interface. The macro can altered slightly to use a drop-down list. The list is populated by an array of strings which must be declared and populated before the dialog definition.

Sub Main()
'DESCRIPTION:Create a 2D frame with supports
Dim arrSupports
arrSupports = Array("Fixed","Pinned")
Begin Dialog UserDialog 600,200,"2D Frame Model" ' %GRID:5,5,1,1
Text 20,20,190,15,"No. of Horizontal Bays:",.Text1
Text 20,45,190,15,"No. of Vertical Bays",.Text2
Text 20,70,190,15,"Vertical Distance",.Text3
Text 20,95,190,15,"Horizontal Distance",.Text4
Text 20,130,190,15,"Support Type",.Text5
TextBox 220,20,130,15,.clmn
TextBox 220,40,130,15,.row
TextBox 220,70,130,15,.ht
TextBox 220,95,130,15,.wdth
DropListBox 240,130,130,50,arrSupports(),.sprt
OKButton 360,165,90,20
CancelButton 490,165,90,20
End Dialog
Dim dlg As UserDialog

The rest of the macro code is unchanged. The selection from the drop-down list is passed as the position in the array selected (e.g., "0" for the first position, "1" for the next position, etc.).

OS. Examples

The following examples have been provided to assist in creating macros using OpenSTAAD in commonly used applications like Microsoft Excel, Autodesk AutoCAD, Microsoft Word.

OS. Simple STAAD.Pro Macro

This example demonstrates a small macro which can be used within STAAD.Pro.

Often, when learning to program, you begin with a program which simply outlines the basic structure of an application, module, or function; typically resulting with a screen message displaying the phrase "Hello World". This example expands upon this to include the foundation from a practical application as well.

Note: Additional examples in this section demonstrate how to poll STAAD data from external programs.

1. Open STAAD.Pro.
2. On the Utilities ribbon tab, select the Macro tool in the Developer group.
The **Macro** dialog opens.

3. Click **Create New**.
   The **New Macro File Name** dialog opens.

4. Type a title of `CreateNewView.vbs` with the following description: *Creates a new view from the selected beams.*
   
   **Note:** You can save the macro file anywhere, so long as the directory has write permission for your user account.

The **STAAD.Pro Script Editor window** (on page 5054) opens with a subroutine title Main.

5. Just after the description, type the following just after the description comment line:

   ```vbs
   Dim objOpenSTAAD As Object
   Dim SelBeamsNo As Long
   Dim SelBeams() As Long
   ```

   This is used to provide some declarations of the objects and variables used in this program.

6. Type the following lines to instantiate the OpenSTAAD object:

   ```vbs
   'Launch the OpenSTAAD Object
   Set objOpenSTAAD = GetObject(,"StaadPro.OpenSTAAD")
   ```

   **Note:** The first line beginning with the apostrophe (') is a comment. It isn't necessary, but it is good practice to add remarks such as this to make your code clear to others (as well as to yourself when you revisit the code at a later time).

7. Type the following lines to set up a logical check for if any beams are selected:

   ```vbs
   'Get no. of selected beams
   SelBeamsNo = objOpenSTAAD.Geometry.GetNoOfSelectedBeams
   If (SelBeamsNo > 0) Then
   ```

   Here, the `GetNoOfSelectedBeams` Geometry function in OpenSTAAD is being used to aid our test. The test is an if... then... else... statement, which continues in the following steps.

8. Type following lines to instruct the program what to do if our statement is true (i.e., there is at least one beam selected).

   That is to create a new view from the active selection using the `CreateNewViewForSelection` View function in OpenSTAAD.

   ```vbs
   ReDim SelBeams(SelBeamsNo) As Long
   'Create a new view
   objOpenSTAAD.View.CreateNewViewForSelections
   ```

9. Type the following lines.

   Since this macro might be run with no beams selected, a message can be provided to the user for some feedback in this instance with the following line:

   ```vbs
   Else
   MsgBox "No beams are currently selected.", vbOkOnly
   End If
   ```

   **Tip:** You could add the message *Hello World*, if you prefer to stick with a more traditional introductory example of programming.
10. Type the following statement to close the instance of the OpenSTAAD object:

   Set objOpenSTAAD = Nothing

This is all it requires to create a macro. Obviously, this particular example really only duplicates the functionality of selecting the **New View** tool in STAAD.Pro. However, it easy to combine other OpenSTAAD functions to automate a series of commonly used features in order to create your own time saving tools.

In this example, for the sake of brevity, the only model entities checked for selection are beams (that is, a new view is only created if beam elements are selected). You could easily expand this to Nodes, Plates, Solids, etc.

<table>
<thead>
<tr>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>The full code for this macro is as follows:</td>
</tr>
<tr>
<td>Sub Main()</td>
</tr>
<tr>
<td>'DESCRIPTION: Creates a new view from the selected beams.</td>
</tr>
<tr>
<td>Dim objOpenSTAAD As Object</td>
</tr>
<tr>
<td>Dim SelBeamsNo As Long</td>
</tr>
<tr>
<td>Dim SelBeams() As Long</td>
</tr>
<tr>
<td>'Launch OpenSTAAD Object</td>
</tr>
<tr>
<td>Set objOpenSTAAD = GetObject(,&quot;StaadPro.OpenSTAAD&quot;)</td>
</tr>
<tr>
<td>'Get no. of selected beams</td>
</tr>
<tr>
<td>SelBeamsNo = objOpenSTAAD.Geometry.GetNoOfSelectedBeams</td>
</tr>
<tr>
<td>If (SelBeamsNo &gt; 0) Then</td>
</tr>
<tr>
<td>ReDim SelBeams(SelBeamsNo) As Long</td>
</tr>
<tr>
<td>'Create a new view</td>
</tr>
<tr>
<td>objOpenSTAAD.View.CreateNewViewForSelections 'SelBeams</td>
</tr>
<tr>
<td>Else</td>
</tr>
<tr>
<td>MsgBox &quot;No beams are currently selected.&quot;, vbOkOnly</td>
</tr>
<tr>
<td>End If</td>
</tr>
<tr>
<td>Set objOpenSTAAD = Nothing</td>
</tr>
<tr>
<td>End Sub</td>
</tr>
</tbody>
</table>

**OS. Microsoft Excel Macro**

A Microsoft Office Excel spreadsheet which can be used to check the capacity of a rectangular concrete beam with the reinforcement already laid out.

This Excel file checks the bending capacity of a rectangular concrete beam with the reinforcement already laid out. The capacity is checked against the maximum sagging moment produced from a series of load cases. The beam is analyzed in STAAD.Pro. The results are extracted and linked into Excel using OpenSTAAD and VBA. A macro name *Exam8* has been created to retrieve the maximum sagging moment along the span of the beam as well as the width and depth of the cross-section of the beam. The maximum sagging moment for each load case is extracted, and the governing moment is the largest compression moment of the maximum moments.

The calculations for checking the capacity are located on the sheet marked “Concrete” while the extraction of the values from STAAD occurs on the sheet marked “STAAD.Pro Output.”

1. Open the file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\OpenSTAAD\Rectangle-Beam.xls.
Note: Excel will warn of macros present in the file. Simply click the Options... button in the Security Warning message area to open the Microsoft Office Security Options dialog, where you will select the Enable this Content option and click OK.

Caution: It is always recommended to do a virus scan on any macro before opening it - including this one!

2. Select the sheet named STAAD.Pro Output and change the values of File Name (cell B14) and Member Number (cell B15) to reflect the file and member number to be considered.

Note: Make sure that there are valid results for the STAAD file selected and they exist in the same directory as the STD file.

3. To run the macro, either:

   select the Macros tool on the Developer tab (Excel 2007 and higher).

   Note: The Developer tab must first be enabled in Excel 2007 and higher. Refer to the Excel help for your version for instructions on how to do this.

   or

   select Tools > Macro > Macros (Excel 2005 and lower).

   or

   press ALT+F8.

   The Macro dialog opens.

   ![Macro dialog in Microsoft Office Excel](image)

   Figure 514: Macro dialog in Microsoft Office Excel

4. Select the macro named Examp8 and click Run to start.

   The values from the STAAD file will automatically be linked into the Excel sheet.

Tip: To see what is going on "under the hood", open the Examp8 macro by clicking Edit from the macro dialog box.
Related Links

- EX. OpenSTAAD Example Files (on page 4311)

OS. Autodesk AutoCAD® Macro

Writing the Member Section Forces at any Distance Along a Member in AutoCAD® 2000 or higher.

OpenSTAAD can also be used to integrate pre- or post-processing data with AutoCAD® 2000 or higher. AutoCAD® 2000 supports VBA macros with its own exposed set of functions. This example automatically calculates the section forces at any distance along a member from a STAAD file and plots the results in a table as a separate layer. A dialog box was created in AutoCAD using AutoCAD VBA forms. The input box will prompt for the STAAD file name, the member number, the load case and the point along the member (expressed as a fraction of the length) where the section forces are to be determined. The macro returns the section forces (by using OpenSTAAD to read STAAD's database) and plots them in a pre-defined table in the DXF file (STDandAcad.dxf).

1. Open the file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\OpenSTAAD\STDandAcad.dwg.

AutoCAD will warn of macros present in the file.

**Caution:** It is always recommended to do a virus scan on any macro before opening it - including this one!

2. Click Enable Macros.

3. To run the macro, either:

   select the RUN VBA Macro tool on the Manage tab (AutoCAD 2009 and higher).

   **Note:** Microsoft Visual Basic for Applications Module is not installed with recent versions of AutoCAD but can be downloaded from the Autodesk website. Refer to the help included with AutoCAD for more information.

   or

   select Tools > Macro > Macros (previous versions of AutoCAD or when AutoCAD 2009 is in AutoCAD Classic mode).

   or

   press ALT+F8.
The Macro dialog opens.

![Macro dialog in Autodesk AutoCAD 2011](image)

**Figure 515: Macro dialog in Autodesk AutoCAD 2011**

4. Select the macro named `STAADSectionForces` and click **Run**.

The **STAAD.Pro Section Forces** dialog opens. A warning also opens informing you that the STAAD File name must be specified.
5. Enter a valid filename, including complete file path, for the current input file open in STAAD.Pro. The Member Number and Load Case drop-down lists are populated with data from the selected model.

6. Select a Member Number and Load Case and specify a Fraction Along Beam.

7. Click OK. The table is updated.

8. (Optional) To populate the table with different results, first clear all contents on the STAAD Results.

**OpenSTAAD Example**

```
<table>
<thead>
<tr>
<th>STAAD File:</th>
<th>Member No:</th>
<th>Distance:</th>
<th>Load Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear Force / Moment</td>
<td>Value</td>
<td>Unit</td>
<td></td>
</tr>
<tr>
<td>FX</td>
<td></td>
<td>Kips</td>
<td></td>
</tr>
<tr>
<td>FY</td>
<td></td>
<td>Kips</td>
<td></td>
</tr>
<tr>
<td>FZ</td>
<td></td>
<td>Kips</td>
<td></td>
</tr>
<tr>
<td>MX</td>
<td></td>
<td>Kips-in</td>
<td></td>
</tr>
<tr>
<td>MY</td>
<td></td>
<td>Kips-in</td>
<td></td>
</tr>
<tr>
<td>MZ</td>
<td></td>
<td>Kips-in</td>
<td></td>
</tr>
</tbody>
</table>
```
### OpenSTAAD Example

<table>
<thead>
<tr>
<th>STAAD File:</th>
<th>Member No:</th>
<th>Distance:</th>
<th>Load Case:</th>
</tr>
</thead>
<tbody>
<tr>
<td>vgs.opj</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Shear Force/Moment</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>FX</td>
<td>58.3245983123779</td>
<td>Kips</td>
</tr>
<tr>
<td>FY</td>
<td>18.1449718475342</td>
<td>Kips</td>
</tr>
<tr>
<td>FZ</td>
<td>0</td>
<td>Kips</td>
</tr>
<tr>
<td>MX</td>
<td>0</td>
<td>Kips-in</td>
</tr>
<tr>
<td>MY</td>
<td>0</td>
<td>Kips-in</td>
</tr>
<tr>
<td>MZ</td>
<td>4183.53515625</td>
<td>Kips-in</td>
</tr>
</tbody>
</table>

**Note:** For more information on how to call AutoCAD commands in VBA, please refer to the AutoCAD VBA Help provided with AutoCAD.

### Related Links
- **EX. OpenSTAAD Example Files** (on page 4311)

### OS. Microsoft Office Security Options

A Microsoft Office Word document is included which contains a partial report for analysis results which can be used as a basis for a customized report document.

The document file includes a macro named Pro which checks for the number of supported nodes as well as the number of load cases and load combinations and reports these back to the document. The document can then report support reactions for the selected node number and load case number.

1. Open an input file within STAAD.Pro.
2. Open the file C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\OpenSTAAD\STAADandWord.doc.

**Note:** Word will warn of macros present in the file. Simply click the **Options...** button in the Security Warning message area to open the Microsoft Office Security Options dialog, where you will select the **Enable this Content** option and click **OK**.

**Caution:** It is always recommended to do a virus scan on any macro before opening it - including this one!

3. Click the **Update Nodes and Load Cases** button.

   The **Node Number** and **Load Case Number** selectors update with the data from the selected file.

4. Select the **Node Number** and **Load Case Number** numbers for which results will be retrieved.

5. Click **Get Results**.

   The support reactions and units update in the table below.
OS. Retrieve Dynamic Output

The following example macro uses several OpenSTAAD output functions to build a mode shape report for the results of a dynamic analysis.

```vba
Option Explicit

Sub Main()
    Dim stdFile As String
    Dim rptFile As String
    Dim Tokens() As String
    Dim staad As OpenSTAAD

    Set staad = GetObject(,"StaadPro.OpenSTAAD")
    staad.GetSTAADFile(stdFile, True)

    Tokens = Split(stdFile,".")
    rptFile = Tokens(0) + ".ModeShapeData.txt"

    CreateModeShapeReport(rptFile, staad)

    Set staad = Nothing
End Sub

Private Function CreateModeShapeReport(rptFile As String, staad As Object)
    Dim I As Integer, J As Integer
    Dim nNodeCount As Long
    Dim nModeCount As Long
    Dim nModeNo As Long
    Dim setOfNodes() As Long
    Dim setOfFrequency() As Double
    Dim modVal(6) As Double
    Dim szName As String
    Dim tblno As Long
    Dim rptno As Long
    Dim idx As Long

    nNodeCount = staad.Geometry.GetNodeCount()

    'Variant GetNoOfModesExtracted();
    nModeCount = staad.Output.GetNoOfModesExtracted()

    Open rptFile For Output As #10
    Print #10, "No of Nodes  = ";nNodeCount
    Print #10, "No of Modes Extracted = ";nModeCount
    Print #10, ""

    If nModeCount > 0 Then
        ReDim setOfNodes(nNodeCount)
    End If
End Function
```
ReDim setOfFrequency(nModeCount)

'Variant GetModeFrequency(Variant varMode, Variant varFreq);
Print #10, "Mode          Frequency (Hz)"
Print #10, "------------------------------------";
For I = 0 To nModeCount - 1
    nModeNo = I+1
    staad.Output.GetModeFrequency(nModeNo, setOfFrequency(I))
    Print #10, nModeNo;Space$(10);Format$(setOfFrequency(I),"Standard")
Next
Print #10,

'Variant GetModalParticipationFactors(Long longMode, Variant varfactorX, Variant varfactorY, Variant varfactorZ);
Dim Participation(3) As Double
Print #10, "Mode    Participation X (%)    Participation Y (%)    Participation Z (%)"
Print #10,
"------------------------------------------------------------------------"
For I = 0 To nModeCount - 1
    nModeNo = I+1
    staad.Output.GetModalParticipationFactors(nModeNo, Participation(1), Participation(2), Participation(3))
    Print #10, nModeNo;Space$(10);Format$(Participation(1),"Scientific"); Space$(14); Format$(Participation(2),"Scientific"); Space$(14); Format$(Participation(3),"Scientific")
Next
Print #10,

'Variant GetModalDisplacementAtNode(Variant varMode, Variant arNode, Variant varModalDisps);
staad.Geometry.GetNodeList(setOfNodes)
Print #10, "Mode  Node         x            y             z"
Print #10, "--------------------------------------------------"
For I = 0 To nModeCount - 1
    nModeNo = I+1
    For J = 0 To nNodeCount - 1
        staad.Output.GetModalDisplacementAtNode(nModeNo, setOfNodes(J), modVal)
        Print #10, Format$(nModeNo,"00");Format$(setOfNodes(J),"000");Space$(5);Format$(modVal(0),"Scientific");Space$(5);Format$(modVal(1),"Scientific");Space$(5);Format$(modVal(2),"Scientific")
    Next J
Next I
Print #10,

'The following function is currently not operational:-
'Variant GetMissingMassParticipationFactors

End If

Close #10

End Function
Index

Numerics
2nd order analysis 2349
3D rendering
  right-click menu 616
A
A1085 93
AASHTO 3828
AASHTO ASD
  verification 3828–3831,
  3833, 3835–3839
AASHTO ASD/LFD
  loading parameters 3134
AASHTO LRFD
  design parameters 1471
  impact factors 3135
  loading parameters 3135
  multiple presence factors
  3135
  verification 3840–3842,
  3844, 3845
AASHTO specifications
  steel design 1463
about
  RAM Connection mode 953,
  954
about STAAD.Pro 28
ABS 2791–2795
acceptance criteria
  criteria 2389
accidental torsion 2694, 2695,
  2698–2702, 2705–2708,
  2721, 2726, 2727, 2729,
  2730, 2732, 2737,
  2739–2743, 2748, 2752–2754, 2758, 2759,
  3017
ACI 2005 294
ACI 318
  beam design output
    1498–1500, 1507
  column design 1518–1520
  column design output
    1514–1516,
    1518–1520
  cracked moment of inertia
    1503, 1504
  design parameters 1481,
    1486, 1487, 1491
  element design 1522–1524
  verification problem
    4254–4256
  verification problem 1 4257,
    4259
  verification problem 3
    4260–4262
  verification problem 4
    4250–4252
  verification problem 5 4253,
    4254
ACI 318-05
  beam briefs 1224, 1226,
    1228
  column briefs 1230, 1232,
    1234
  RC Designer 229
ACI 318-14
  bar positions 1497
  beam design 1495, 1496
  column design 1509
  metric 97
  output 1498–1500
  verifications 4263
ACI 318-98 1086
ACI 318-99
  beam design brief
    1086–1088
  beam design principles
    1086–1088
ACI 318M-05
beam briefs 1224, 1226,
  1228
column briefs 1230, 1232,
  1234
ACI 318R-98 1086
ACI Beam Design Brief
  1218–1220
ACI Column Design Brief 1092,
  1221–1223
ACI Column Design Principles
  1092
actions
  deformation controlled
    2381
  force controlled 2381
ActiveX component error 5059
ADAPT-Builder 311, 335
Add New Pushover dialog 3049
adding pushover loading 864
advanced analysis
  nonlinear cable members
    2332
Advanced Concrete Design 1352
advanced nonlinear cable
  analysis 2358–2361
advanced slab design mode
  1353
advanced solver 267, 2404,
  2405
AIJ
  seismic definition 285
AIJ (Japanese) 2605–2607
AIJ 2005 228
AIJ Beam Design Brief
  1062–1064, 1239, 1240
AIJ Beam Design Principles
  1062–1064
AISC
  effective length factors 327
  unified code 281, 294
  verification examples 3847
AISC Steel Design Guide 2 1432
AISC steel tables 1368–1370
AISI
design parameters 1476
AISI steel sections 1474
AITC
elements 1541, 1542
AITC 1984
example 1530, 1531
AITC 1985
design operations 1527, 1528
design parameters 1533, 1534, 1537
AITC 1994
design operations 1527, 1528
design parameters 1533, 1534, 1537
member selection 1538
Al clad 1544, 1550
Algerian 2546–2548
allowable stresses
AISC 9th Ed 1406
AITC 1985 1528–1530
AITC 1994 1528–1530
aluminum
design codes 1362
aluminum design
American 1543
code checking 1363
commands 1363
designing members 1363
generating take off 1364
initiating 1363
NS 3472 NPD 2050
selecting members 1363
specifications 2850
specifying parameters 1363
aluminum table 2969
American codes 1366
American Transmission Tower Code 1558
analysis
buckling 263, 932, 2804, 2805
changing 934
compression only 2356
direct analysis 292, 2352, 2806, 2807
dynamic 2362
dynamic, verification problems 3592
facilities 2346
geometric nonlinear 2355, 2830, 2832
harmonic response 2374
imperfection 932, 2355, 2832
linear elastic 928, 2796
modal 266, 267
multilinear springs supports 2356
multiple 2835, 2836
multiple analyses 2401
nonlinear cable 931, 2357, 2800, 2803
nonlinear cable advanced 2358–2361
nonlinear static, verification problems 3583
nonlinear truss 2357
P-Delta 262, 928, 1630, 2349
P-Delta options 2797–2800
P-Delta 262
perform nonlinear 930
performing 942
post-analysis output 939
pre-analysis commands 936
pushover 2376, 2386
response history 2767–2769
response spectrum 2371, 2372, 2687
response time history 2373, 2374
second order 2349
secondary 2400
static 2796
steady state 333, 2374
steady state and harmonic 2807
stiffness 2346–2348
tension only 2356
time history 2373, 2374
types 928
analysis considerations
concrete design 1521, 1522
analysis results
colors 1132
printing 2840, 2841, 2844, 2845
analyze 4950
animation 2213
ANSI/AISC N690 codes 1575
ANSI/AISC N690-1994 174
API 1562
APL Apollo 91
application window layout
ribbons 2262
archives
creating 2272
extracting 2273
opening 2272
archiving 326
area loads 830, 2338, 2665
Arnoldi/Lanczos method 2363, 2364, 2413, 2414, 2419, 2422
AS 1170 1634, 1635
AS 3600
RC Designer 229
AS 4100-1998
verification problem 1 3640, 3642
verification problem 2 3643, 3644, 3646
verification problem 3 3647, 3649
ASC steel decks 695, 2918
ASCE 7
snow loads 2345
soft story 132
soft story check 937
wind load 836
wind load intensities 2623–2626, 2628, 2629
wind loads 3039–3042
wind pressure 3038, 3039

STAAD.Pro 5090 User Manual
ASCE 7-02
snow loads 851, 2785
snow load 344–346
wind load 337, 339, 341, 343
ASCE 7-05 848
See also IBC 2006
ASCE 7-10
response spectra 94
seismic loads 94, 115
wind load 3574, 3577–3579
ASCE 7-16
seismic loads 87
ASCE 7-2010
add wind intensity 140
ASCE 7-95
soft story 2526–2528
ASCE Pub 52 1558, 1559
ASME NF
load combinations 2791–2795
ASME NF 3000
verifications 4122
ASME NF codes 1588
assign
profile specification 2470
assigning material constants 2501
associating
ProjectWise projects 2274
ATC-40 2376
austenitic stainless steel 1575
Australian
circualar hollow sections 204, 205
cold-formed steel hollow sections 204, 205
Duragal 204, 205
Galtube 204, 205
OneSteel 204, 205
profiles 204, 205
rectangular hollow sections 204, 205
square hollow sections 204, 205
Australian codes 1627
auto load rules editing 3035–3037
auto save enabling 2271
automatic load combinations
dialog 3033, 3034
automatic spring support
generator
foundations 2517–2519
AutoPipe
exporting to 2289
AutoPIPE
neutral file 882
autosifting eigen vectors 2363, 2364
AVI files 2213
axial compression
AISC LRFD 1453
axial tension
AISC LRFD 1453
axial-only 2330
axially loaded members
design 1745, 1746, 1758
batch design
performing 942
BD21/01
loading parameters 3133
BD21/01 Annex D
impact factors 3133
loading parameters 3133
beam design
ACI 318 1503
ACI 318-14 1495, 1496
beam end forces 3130, 3131
beam force details 3157
beam-girder identification
dialog 3174
beams
bending diagrams 2216
curved 646
design calculations 4969
designing 4964
display dimensions 893
elevations 4967
incidences, changing 892
insert nodes into 2906
inspect properties 49
labels 617
load cases 4963
load combinations 4963
merge dialog 2909
modifying 4964
overlapping 698
results 2214
save project 4970
schedules 4967
set attributes 643
shear diagrams 2216
specifications 2962–2966
split groups 4964
start and end 634
start design 4961
steel design utilization
ratios 2216
stress contours 2215
stretch dialog 2908
top elevation 4964
BEAVA
using 884
See also Bridge Deck workflow
Belgian NA
   Eurocode 3 145
bending
   AISC LRFD 1454
bending stress
   AISC 9th Ed 1406
Bentley Cloud Services 2878
Bentley Rebar 297
beta angle
   assigning 794
   dialog 2978
   reference point 324
   reference point, dialog 2979
beta angles 256
BFS NE 1993-1-1 1792
biaxial bending
   AISC LRFD 1454
block fireproofing 2482–2484
bond lengths 1081–1083
bookmarks
   adding 2257
   removing 2257
boundaries 1165
boundary conditions
   torsional 1407–1409
      See also supports
braced frames 2386
braces
   curly 2407, 2408
bridge codes 884
bridge deck workflow
   opening 885
Bridge Deck workflow 883, 884
bridge decks
   creating 885
   defining 885
   select plates 3124
   selecting 3130, 3131
bridge standards
   selecting 886, 3130, 3131
briefs
   design 1010, 1011
British codes
   BS 5950 1655
   BS 5400
      loading parameters 3132
   BS 5950
calculation sheets 332
weld type 1661, 1666
BS 5950-1-2000
verification problem 1
   3787–3789
verification problem 2
   3794–3796
verification problem 3
   3801–3803
verification problem 4
   3808–3810
verification problem 5
   3814–3816
verification problem 6 3819, 3820
verification problem 7
   3823–3825
BS 8110
   RC Designer 333, 334
BS EN 1993-1-1 1776, 1785, 1792
BS4360 1661, 1666
BS8110 1081
BS8110 Beam 4940
BS8110 Beam Design 4940
BS8110 Beam Design Brief
   1255–1257
BS8110 beam design briefs
   1081–1083
BS8110 beam design principles
   1081–1083
BS8110 Column 4944
BS8110 Column Design 4944
BS8110 Column Design Brief
   1257–1259
BS8110 recommendations 1085
BS8110 Slab Design Brief 1085,
   1260, 1261, 1294–1296
BS8110 Slab Design Principles
   1085
buckling analysis
   advanced solver 2354
   eigen method 2353
Building Modeler
   example, analysis model
   4922
building planner
add beam 904
add beams automatically 906
add column 902
add columns automatically 903
add irregular slab 900
add rectangular slab 899
automatically add beams 906
automatically add columns 903
beam continuity 908
beam design 926
beams, creating 904
cantilever beams 909
column design 927
column orientation 917
column sizes 917
columns, adding 902
create frame 911
create shear wall 912
design parameters 910
design, beams 926
design, columns 927
design, slabs 926
DXF import 897edit plan 898
edit slab 901
examples 4309
finalize plans 911
frames, creating 911
generate STAAD.Pro model
   923
import plan 897
load combinations 923
loads, combinations 923
loads, seismic 921
loads, wind 920
member releases 915
new plan 895
plans, creating 895
plans, importing 897
PlanWin import 897
seismic parameters 921
shear walls, creating 912
slab design 926
START DECK DEFINITION 2473–2475
START GROUP DEFINITION 2440–2442
STEEL TAKE OFF 2856, 2857
SUBSTITUTE 2439, 2440
SUPPORT DISPLACEMENT 2683, 2684
SURFACE CONSTANTS 2508, 2509
SURFACE DIVISION 2846
SURFACE PROPERTY 2488
TEMPERATURE LOAD 2682
TIME LOAD 2767–2769
typing 2255
UNIT 2411–2413
VDB 2823, 2824
commenting
STAAD input language 2407, 2408
comments
adding 2255
compatibility 5050
complete quadratic combination method 2372
composite beam design
AISC 9th Ed 1420
AISC LRFD 1458, 1459, 1462, 1463
composite beams
change properties 695
decks 2473–2475
composite damping
springs 2511
composite decks
assigning properties 695
change beam properties 695
create from perimeter beams 694
deleting 694
loading 831
specify rib direction 694
composite slab
geometry 2976
compression stress
ASIC 9th Ed 1406
compression-only members
specifying 798
compression-only springs 251, 2522–2524
compression-only supports 2986
concentric braced frame 2386
cracking design codes 1000, 1001
cracking design
ACI 318 1478
ACI 318 verification problems 4250
Advanced Concrete Design 1352
AIJ 1991 1969
AS 3600 1627
beam design 1003
BS8007 1680
BS8110 1655
CSA A23.3-94 1696
earthquake checks 1018
Eurocode 2 1742
initialing 1002
interactive 1005
IS13920-1993 1901
IS13920-2016 1955
IS456 1887
NCT 1987 2002
Russian 174
SABS-0100-1 2176
section types 1480
shear 1497
SNiP 2.03.01-84* 174
SP 63.1330.2012 2164
specifying parameters 1002
take off 1004
UK example 10 4716, 4719, 4720
UK example 8 4698, 4701, 4702
UK example 9 4428, 4431, 4432
verification examples 4250
Concrete Design Command 2860
Concrete Design Specifications 2859
Concrete Design Terminator 2861
Concrete Design workflow 1005
cracking member design
entering 1211
Concrete Take Off Command 2860
configuration
key files 289
CONNECT Advisor 106, 2278
CONNECT Advisor dialog 2279
CONNECT Licensing 90
CONNECTED projects
disassociating 2275
See also ProjectWise projects
CONNECTION Client updates 2283
connection design
HBBB 969
HCBB 969
connection design workflow
settings dialog 977
connection designs
basic connections 967
basic connections dialog 3166
connection sets 954
export to report 971
gusset connections 968
gusset connections dialog 3171
settings 966
smart connections 968
smart connections dialog 3169
connection input table 974–976
connection material database table 3176, 3177
Connection pad 980
connection tags
Assign dialog 3183
Check dialog 3100
checking 985
command syntax 2537
creating 982
deleting 983
eexample 4933
New dialog 3185
removing assignment 984
sample XML file 994
XML Schema 986, 988–993
connection templates
add custom template 973
create custom 972
custom 972
connections
British sections 231
piping, manually specifying 880
constants 788
constrain
to slab X 1163
continuing commands 2407, 2408
contour fireproofing 2482–2484
coordinate systems
Cartesian 2296, 2425–2428
cylindrical 2296, 2425–2428
definition 44, 45
local 2297
reverse cylindrical 2296, 2425–2428
coordinates
Z up 323
copy
along arc 702
along line 702
picture 1117
See also translational repeat
corner releases 807
cover plates 118, 2976
CP 65 1071
CP 65 Beam Design Brief 1072, 1073
CP 65 Column Design Principles 1074
cqc , See complete quadratic combination method
crack condition
BAEL 1252–1254
cracked moment of inertia 1503, 1504
cracked section properties
assigning 802
cracked sections 249
cracking
member properties 249
create user provided table 2974
creating groups 704
creating new views 627
CSA timber 2857
CSA-A23.3-04
beam design brief 1262, 1264
CSA-A23.3-10
beam design principles 1021–1023
column design brief 1265, 1267, 1268
column design principles 1024
curly braces 2407, 2408
current units
changing 41
curved beams
add graphically 646
curved members 2329, 2475–2477, 2480, 2481
custom
catalog sections 787
custom vehicles 3138
cut 1117, 2882
cut-off
frequency 2539
mode shapes 2539
time 2539
cutting sections 623
damping
composite 2367
define for dynamics 3059
modeling 2366
springs 2511
damping ratio 3059
damping ratio, modal 2503, 2505–2508
dashes 2407, 2408
data area 61
data files 2251
data separator 2407, 2408
DD ENV 1742, 1743
DD ENV 1993 1745, 1746
define
starting load 2648, 2649
define hinge property 2825
defining
bridge deck 885
push load pattern 2823
pushover data 862
pushover data, members 862
roadways 885
solution control 865
definitions
load systems 2540
material constants 2501
moving loads 2541–2543
snow loads 2641
static force procedures 2544
time history load 2630, 2639, 2640
wind loads 2623–2626, 2628, 2629
deflection check
steel design 1419
delete members 2445, 2446
demand spectrum
generation 2390, 2391
input 2390
density 2503, 2505–2508
deprecated commands
printer plot 2849
problem statistics 2795
drift
assigning 800
ductility design
ACI 318-14 1496
duplicates
beams 3096
plates 3096
Duragal 204, 205
DXF
export 2877
exporting to 2288
import 2877
importing as geometry 2286
DXF files
import dialog 2878
dynamic analysis
miscellaneous settings 2538
verification problems 3592
dynamic loading specification 2686
dynamic results 2217
dynamic wind loads 839

E
E-mail 52
earthquake checks
Concrete Design 1018
earthquake collapse check 1105
Earthquake mode
Check Regularity in Plan dialog 2244
Earthquake Elevation Criteria dialog 2245
elevation regularity 2245
Elevation Regularity table 2245
Floors dialog 2244
Soft Story table 2244
Story Stiffness table 2244
Torsional Radius Check table 2245
Earthquake workflow
EC8 Stiffness page 2243
elevation irregularities 2243
opening 2241
pages 2243

plan irregularities 2242
plan regularity 2244
stiffness design 2244
using 2240

EC2
2004 1044
eccentricity 2694, 2695,
2698–2702, 2705–2708,
2721, 2726, 2727, 2729,
2730, 2732, 2737,
2739–2743, 2748,
2752–2754, 2758, 2759

ECCS 203 1042
ECCS 203 Beam Brief
1301–1303
ECCS 203 Beam Design Brief
1043, 1301–1303
ECCS 203 Beam Design
Principles 1043
ECCS 203 Column Brief
1304–1306
ECCS 203 Column Design Brief
1044, 1304–1306
ECCS 203 Column Design
Principles 1044

editing 2255
editor 2251
effective length factors
auto-calculation 327

EHE 1076
EHE Beam Brief 1307–1312
EHE Beam Design Brief 1076,
1077, 1307–1312
EHE Beam Design Principles
1076, 1077
EHE Column Brief 1313–1317
EHE Column Design Brief
1313–1317
EHE Column Design Principles
1077
eigen vectors
Arnoldi/Lanczos method
2413, 2414, 2419, 2422
autoshifting 2363, 2364
eigenproblem
solution 2363, 2364

STAAD.Pro 5098
User Manual
EN10219-2 138, 139
end releases 799
end run 2862
envelope
design loads 976
Envelope Page 1202
envelopes
load 2837, 2838
envelopes, load 318, 320
equivalent slenderness 1659–1661
equivalent lateral force
IBC 2006 258
error message
ActiveX component 5059
errors
checking 2256
Eurocode
combined stress 1770, 1771
load combinations 870
steel design 1745, 1746, 1757, 1758
Eurocode 2
preliminary design 1018
Eurocode 2-2004
RC Designer 230
Eurocode 3
Belgian national annex 145
DD ENV 1993 1742
EN 1993 1754
Finnish national annex 160
French national annex 178
Germany National Annex 121
Malaysian national annex 134
Polish national annex 160
slender box sections 227
steel grades 226
torsion design 158, 159
UK national annex 178
UK National Annex example 4725, 4728, 4729
Eurocode 5 1856, 2857
Eurocode 8
earthquake checks 2240
perform checks 1018
response spectra, adding 849
response spectrum 285
response spectrum example 4725, 4728, 4729
European codes 1742
European Codes 1742, 1758
example
EXAMPPUSH01.STD 5039
single moment frame 5039
examples
AITC design 1541, 1542
American design 4312
British design 4627
BS 5400 Part 2 5033
BS8110 design 4936
building planner 4309
CIS/2 4310
connection tags 4933
macros 4310
modeling 4909
OpenSTAAD 4311
physical models 4311
steel design 4925
UK example 1 4627, 4632,
4633
UK example 10 4716, 4719, 4720
UK example 11 4725, 4728, 4729
UK example 12 4735, 4738, 4739
UK example 13 4743, 4746, 4747
UK example 14 4749, 4752, 4754
UK example 15 4760, 4763, 4765
UK example 16 4774, 4778, 4779
UK example 17 4781, 4785, 4787
UK example 18 4789, 4791–4793
UK example 19 4795, 4797, 4798
UK example 2 4645, 4648, 4649
UK example 21 4806, 4809, 4810
UK example 22 4813, 4816, 4817
UK example 23 4822, 4825, 4826
UK example 24 4833, 4837, 4838
UK example 25 4847, 4850, 4851
UK example 26 4855, 4859, 4860
UK example 27 4862, 4866, 4868
UK example 28 4873, 4878, 4879, 4885
UK example 29 4887, 4892, 4893
UK example 3 4663, 4666, 4667
UK example 4 4670, 4675, 4676
UK example 5 4683–4685
UK example 6 4687, 4690
UK example 7 4693, 4695, 4696
UK example 8 4698, 4701, 4702
UK example 9 4707, 4710, 4711
US example 1 4313, 4317, 4318
US example 10 4436, 4439, 4440
US example 11 4443, 4447, 4448
US example 12 4454, 4457, 4458
US example 13 4462, 4465, 4466
US example 14 4468, 4471, 4473
US example 15 4480, 4483, 4484
US example 16 4493, 4496, 4497
US example 17 4500, 4504, 4505
US example 18 4509, 4511–4513
US example 19 4515, 4517, 4518
US example 2 4363, 4366, 4367
US example 20 4522, 4524, 4803, 4804
US example 21 4526, 4529, 4530
US example 22 4533, 4536, 4537
US example 23 4541, 4544, 4545
US example 24 4552, 4556, 4557
US example 25 4566, 4569, 4570
US example 26 4574, 4577, 4579
US example 27 4580, 4584, 4586
US example 28 4591, 4596, 4597
US example 29 4605, 4610, 4611
US example 3 4383, 4386, 4387
US example 4 4390, 4394, 4395
US example 5 4403–4405
US example 6 4407, 4410
US example 7 4413, 4415, 4416
US example 8 4418, 4422, 4423
US example 9 4428, 4431, 4432
wall-slab connection 4909
export
Bentley Rebar 297
CIS/2 194, 196, 197, 2877
DXF 2877
QSE 2877
SACS 3107
STAAD model to AutoPIPE 882
to AutoPIPE 881, 882
VRML 2877
Export to Drawing dialog 1188
exporting
AutoPipe 2289
CIS/2 file 2289
DXF file 2288
to SACS 2289
exporting connection designs 971
extrusions 1544, 1550

F

FEMA 365 2376
file extensions 1007
file types 1007
fillet
  free sketch 766
find
  search methods 2258
finite element meshing
  wall-slab interface 348
finite elements
  plates 2308
  shells 2308
  solids 2319, 2321
  surfaces 2322
Finnish NA
  Eurocode 3 160
fire proofing
  assigning 803
fireproofing
  applying 2482–2484
  block 2482–2484
  contour 2482–2484
fixed end loads 829
fixed supports
  assigning 812
fixed-end member loads 2340, 2683
flexural design strength
  AISC LRFD 1453
floor diaphragms 2526–2528, 2961
floor load 2672–2677
floor loads
  composite decks 831
floor response spectrum 199, 206
floor slabs 1008
floor spectra
  generating 935
  results 2218
floor spectrum
  generate 2833, 2834
floor structures 2409, 2410
floor vibrations
  generate report 2219
  theory 2221
fonts 2268
footings 2985, 4949
force envelope
  printing 2848, 2849
force envelopes 2401
forcing function 2373, 2374
forming members
  rules 1009
forming slabs
  rules 1010
Foundation Design
  exporting 1360
  exporting selection 1361
  Foundation Design dialog 1361
  opening 1360
  using 1359
foundations
  subgrade modulus 2517–2519
  supports 2985
FPS 2295
frame element hinge 2382
frames table 979
French
  codes 1872
  steel design 1872
French NA
  Eurocode 3 178
frequencies
maximum number 2409, 2410
frequency
adding to load case 834
fully restrained moment frame 2386
Function
  Return Value 5049
fundamentals 31

G
Galtube 204, 205
gamma angle
curved members
  2475–2477, 2480, 2481
GB 1591 1661, 1666
GB50010
  Beam Brief 1318–1320
GB50010-2002
  specifications 1025, 1026, 1029
GB50011-2001 179
general sections
  creating 735, 739
generate floor spectrum 199
generated mesh
  parametric models 2434
generating
  influence surfaces 885
loads on roadway 886
generation
  loads 2771
geometric axis
  , See RAngle
gеometric nonlinear analysis
  2355
gеometric nonlinear analysis
  2830, 2832
gеometric nonlinearity 2381
gеometric stiffness matrix 2349
German codes 1880
getting help 81, 82
getting started 953, 2251, 4949
global coordinate systems
  related to local 2301, 2302
global ranges 2409
global support specification
  2514, 2515
go to line 2256
goto row 1120
Graph Colours dialog
  displays 1176, 1177
graphical view window 615
gravity loading
  pushover analysis 2387
grid marks
  Steel AutoDrafter 2231
grid tool 236, 239, 240
grids
  adding 639
  import DXF 641
  importing 641
groups
  adding to existing 709
  creating 704
  specification 2440–2442
gusset connections
  dialog 3171
  selecting 968

H
H15-44 3138
H20-44 3138
hard drive 28
harmonic distributed load
  3630–3632, 3634, 3636
harmonic force loading 2813, 2814
Harmonic Ground Motion
  Loading 2812
harmonic loading function 2373, 2374
harmonic output frequencies
  2809
harmonic response 2374, 2807
harmonic results
  printing 2815
HBBB
  connection design 969
HCBB
  connection design 969
help
  browse contents tab 25
  browser 106
  opening 24
  searching 25
  using 24
  using the index 25
  view next topic 25
  view previous topic 25
hinge properties
  defining 2388, 3053
hinges
  formation 2391
  frame element 2382
  manually assign 863
  manually define 863
  plastic, location of 2391
  unloading 2391
  unloading, method of 2392
horizontal brace connections 98
horizontal response spectrum
  2390, 2391
HS15-44 3138
HS20-44 3138
hydrostatic loads 833

I
I-shaped beams
  ACI 318 1505
IBC
  verifications 3422
IBC 2000
  response spectrum, adding 844
IBC 2006
  equivalent lateral force 217, 219, 220
  equivalent lateral force 258
  response spectrum 228, 269
  response spectrum, adding 848
IBC 2012
  seismic load definition 2596, 2598–2600
IBC 2015
seismic load definition 2600, 2602
IBC 2018
  seismic load definition 2603–2605
IBC response spectra 3027
ignore 2423
ignore inplane rotation 2498, 2967
impact factors
  AASHTO LRFD 3135
  BD21/01 Annex D 3133
imperfection
  analysis 932
  member specification 800
imperfection analysis 2355, 2832
imperfection information 2511–2513
imperfection member 2966
import
  CIS/2 194, 196, 197, 2877
  DXF 2877
  from AutoPIPE 878
  QSE 2877
  Stardyne 2877
importing
  CIS/2 2287
  DXF 2286
inactive members
  example 4390, 4394, 4395, 4670, 4675, 4676
inclin
  joint loads 2651–2653
inclined loads 824
inclined supports 817, 2516, 2517, 2984
Indian codes 1887
influence diagrams
  scale 3136
  selecting 3136
influence surfaces
  generating 885
input file
  creating 2254
  opening 2254
input files 49, 2253
input generation 2294
input instructions 2404
input language conventions 2405
input specification 1530, 1531
input width 2413
insert nodes
  multiple members 697
  single member 697
installation 30
  Instantiate 5048
  Integrated Structural Modeling 2269
inter-story drift . See story drift
interoperability 2251
intersection
  beams 698
introduction
  STAAD input instructions 2404, 2405
IRC
  loading parameters 3135
  multiple presence factor 3135
irregular modes of oscillation 2533
irregularities
  IS 1893 2016 2531–2534
IS 1893
  Seismic Load 2576, 2577, 2579–2581
  Floor Level Definitions 2581, 2582
  Response Spectrum 2721, 2726, 2727, 2729, 2730, 2732, 2737, 2739–2743
response spectrum, adding 846
response spectrum update 286
seismic irregularities 937
soft story 2526–2530
soft story check 937
story drift 2721, 2726, 2727, 2729, 2730, 2732, 2737, 2739–2743
verification 3478
IS 1893 - 1984
  static seismic load definition 2576
IS 1893 2002
  story drift 133
IS 1893 2016
  cracked section properties 2485, 2486
  reduced section properties 2485, 2486
  response spectra 95
  seismic irregularities 2531–2534
  seismic load, static 2582, 2583, 2585, 2587
  seismic loads 94
  soft story check 95, 96
IS 456
  element design 1900, 1901
IS 801 87
IS13920-2016
  beam design 1956, 1957
  column design 1957
IS1893 2016 84
IS456 Beam Brief 1325–1327
IS456 Beam Design Brief 1325–1327
IS456 Column Design Brief 1059, 1060, 1328–1330
IS456 Column Design Principles 1059, 1060
ISM
  application data 2271
  menu items 2876
  purpose 2270
  sync tools 2269, 2270
  what is it? 2270
ISM 1893 2002
soft story 132
isotropic material
creating 788
isotropic materials 2987

J
Jacobian 893
Japanese codes 1969
Japanese seismic load
2605–2607
Jindal steel 163
job information 1109, 2424
Job Information dialog 1202
joint coordinates
printing 2840, 2841, 2844,
2845
specifications 2425–2428
joint loads 824, 2337,
2651–2653
See also nodal loads
joints
coordinates 2425–2428
coordinates, printing 2840,
2841, 2844, 2845
element load 2661, 2662
limit of generated 2435,
2436, 2438
master-slave 2336
maximum number 2409,
2410
redefinition 2439, 2440
special selection, dialog
3166
joist girders 2326–2328
JPSL 163

K
key files 289
keyboard shortcuts 78, 2268
Kg matrix 2349

L
labels
individual 622
support reactions 2213
dialog 3080
load specifications
  pmembers 2655
load systems
  definitions 2540
load types
  display options 291
load-depantant Ritz vectors
  starting vectors 865
loadcases 1008
loading 819
loading parameters
  AASHTO ASD/LFD 3134
  AASHTO LRFD 3135
  BS 5400 3132
  BS21/01 3133
  BS21/01 Annex D 3133
  dialog 3130, 3131
  IRC Chapter6 3135
loads
  selfweight 2685, 2686
  area loads 2338
  ASCE 7 snow 2345
  ASCE 7 wind 3039–3042
  auto load rules 3035–3037
  definitions 2540
  dialog 2994
  display colors 3137
  displaying 631
  elements 2342
  envelope 877
  envolopes 318, 320
  Eurocode 8-1994 response
    spectrum 2710,
    2714, 2715
  fixed-end member loads 2340
  fixed-end specification 2683
  floor 2672–2677
  generating on roadway 886
  generation 2771
  ground motion 2767–2769
  harmonic force loading 2813, 2814
  IBC 2006 response
    spectrum 2743,
    2747, 2748
IBCA 2012 response
  spectrum 2748,
  2752–2754
IBCA 2015 response
  spectrum 2754,
  2758, 2759
IS 18932-2002 response
  spectrum 2721,
  2726, 2727, 2729,
  2730, 2732
IS 18932-2016 response
  spectrum 2732,
  2737, 2739–2743
joint loads 2337
load lists 3080
mass modeling 876
member loads 2337, 2653,
  2655
moving 853
moving loads definition
  2541–2543
moving loads generation
  2343
notional 2785–2787
NRC 2005 response
  spectrum 2694,
  2695, 2698–2701
NRC 2010 response
  spectrum 2702,
  2705–2708
oneway 2666, 2667, 2669,
  2671
post-tensioned members 2340
poststress 2340, 2678, 2679
prestress 2340, 2678, 2679
prestress, example 4407,
  4410, 4687, 4690
prestressed members 2340
primary load cases, dialog 2771
pushover 861
reference 873
repeat load case 872
repeat loads 2770
response spectrum 2688,
  2692–2694
seismic 840
  seismic definition 3017
  seismic, RPA (Algerian)
    2546–2548
selfweight 2685, 2686
SNiP II-7-81 response
  spectrum 2759,
  2762, 2763
SNiP wind loading 837
snow 851, 2641
specifications 2650
static seismic forces 2544
strain 2341, 2682
support displacement 2683,
  2684
  support displacement,
    example 4403–4405,
    4683–4685
support displacemnets 2342
surface elements 2663,
  2664
temperature 2341, 2682
time history 855, 2630,
  2639, 2640
time varying 2767–2769
transfering to STAAD.Pro
  887
types 47, 48
UBC 1985 seismic 2617,
  2619, 2620
UBC 1994 seismic 2617,
  2619, 2620
UBC 1997 seismic
  2620–2623
UBC seismic 2343
wind 835, 2345
wind definition 835
wind load definition
  exposure 837
  wind load, applying 838
local coordinate systems
  related to global 2301, 2302
location of plastic hinges 2391
LRFD fundamentals 1452
M
macro
add to menu 5051
create 5051
break points 5074
initialize OpenSTAAD 5072
testing 5074
macros
add option controls 5067
add to user tools 5075
adding dialog buttons 5069
adding labels to dialogs 5064
adding text fields to dialogs 5064
create form 5062
creating new 5051
dimension variables 5070
example 5075, 5078
example files 4310
Export to AutoPipe 882
generate members 5073
get user input 5071
run linked 5052
STAAD.Pro Script Editor 5054
user dialogs 5062
Malaysian NA
Eurocode 3 134
manuels
International Codes Manual
27, 28
27, 28
mass irregularities 2533
mass modeling
reference load tables 2645
reference loads 2642, 2643,
2646, 2647
weight tables 2644, 2645
master-slave
joints 2336
specifications 2525, 2526
mat foundations 2985
material constants
assigning 2501
definition 2501
dialog 2977
members 2503, 2505–2508
material definitions
creating 788
creating orthotropic 791
material nonlinearity 2381
material properties
printing 2840, 2841, 2844,
2845
material take-off
Steel AutoDrafter 2238
materials
assigning 792
definitions, creating 788
isotropic 788, 2987
orthotropic 2987
mats
elastic 252
plate 252
member
compression 2330
displacements 2400
tension 2330
member attributes
define 2535, 2536
member cable specification 2492
member compression
specification 2492
member end forces 2395, 2398
member end offsets
example 7 4413, 4415,
4416, 4693, 4695, 4696
member end releases
assigning 800
member forces
intermediate sections 2400
printing 2840, 2841, 2844,
2845
member groups 1012
member incidences
specification 2428–2430
member information
printing 2840, 2841, 2844,
2845
member loads
fixed-end 2340
poststress 2340
prestress 2340
specification 2653, 2655
member offsets 2334, 2499,
2500
member properties
beams 1006, 1007
built-in property tables
2459, 2460
castellated beams 1428
columns 1006, 1007
cracked section 249
equivalent 2472
printing 2840, 2841, 2844,
2845
shear area 2322
specification 2459, 2460
steel members 1367
tapered sections 2468, 2469
user-defined 2459, 2460
member query
physical members 245
member release specification
2488–2490
member releases 2329,
2488–2490
member selection
AISC LRFD 1458
AITC 1994 1538
optimization 1419
member specifications
cable 2963
compression-only 2964
dialog 2962–2966
fire proofing 2965
imperfection 2966
inactive 2965
offset 2964
property reductions 2962
releases 2962–2966
tension-only 2962
truss 2964
member stresses
printing 2840, 2841, 2844,
2845
exporting 2288
importing 2286
open existing 34
modes
advances slab design 1353
Concrete Slab 1212
Earthquake 1216
maximum number 2409, 2410
piping 878
See also workflows
modulus of elasticity 1638, 1639
moment magnification
ACI 318-14 1511
moment of inertia
cracked 1503, 1504
move origin
dialog 2903
moving
dialog 2902
moving load generator 854
moving loads
defining 853
definition 2541–2543
example 4454, 4457, 4458, 4735, 4738, 4739
generation dialog 3031
generator 2343
MS NE 1993-1-1 1792
Multi-RC 1081–1083
multilinear spring supports 2356, 2520, 2521, 2985
multiple analyses 2401
multiple presence factor
AASHTO LRFD 3135
IRC Chapter 6 3135
multiple structures 887
MultiREBAR drawing 1188

N
N690 codes 1575
national annex
Singaporean 1776, 1785, 1792
Belgian 1776, 1785, 1792
British 1776, 1785, 1792
Dutch 1776, 1785, 1792
Finnish 1776, 1785, 1792
French 1776, 1785, 1792
German 1792
Malaysian 1792
Norwegian 1776, 1785, 1792
output 1790
Polish 1776, 1785, 1792
Swedish 1792
National Annex 1776, 1785
National Application Documents 1742, 1743
natural frequencies
verification problem 3611–3620
natural modes
verification problem 3608–3610
verification problem, frame 3621–3625
natural torsion 130, 2840, 2841, 2844, 2845, 3017
navigating
go to line 2256
NBN EN 1993-1-1 1776, 1785, 1792
NEN-EN 1993 1776, 1785, 1792
NORSOK N-004 157, 158
Norwegian codes 2046
notional loads
adding 852, 3028, 3029
NRC
verification 3553
NRC 1995
seismic load definition 2548–2551
NRC 2005
seismic code 283
NS-EN 1993 1776, 1785, 1792
NS3473 1064
NS3473 Beam Design Brief 1331–1336
NS3473 Beam Design Principles 1065, 1066
NS3473 Column Design Brief 1337–1341
NS3473 Column Design Principles 1066
NTL file (AutoPIPE) 882
nuclear power plants 198 labels 617
loads, adding 824
master, assigning 810
master, displaying 638
master/slave 2960
orphans, removing 892
results 2214
specifications 2960, 2961
nomenclature 5049
nonlinear analysis
cable 2357
geometric 2355, 2830, 2832
truss 2357
nonlinear cable analysis 2357
nonlinear cable members 801, 2332
nonlinear static analysis
verification problems 3583
nonlinear truss analysis 2357
nonlinearity
geometric 2381
material 2381
types 2381
nonsymmetric sections 1420
Norsok N-004 157, 158
Norwegian codes 2046
notional loads
adding 852, 3028, 3029
NRC
verification 3553
NRC 1995
seismic load definition 2548–2551
NRC 2005
seismic code 283
NS-EN 1993 1776, 1785, 1792
NS3473 1064
NS3473 Beam Design Brief 1331–1336
NS3473 Beam Design Principles 1065, 1066
NS3473 Column Design Brief 1337–1341
NS3473 Column Design Principles 1066
NTL file (AutoPIPE) 882
nuclear power plants 198 labels 617
loads, adding 824
master, assigning 810
master, displaying 638
master/slave 2960
orphans, removing 892
results 2214
specifications 2960, 2961
nomenclature 5049
nonlinear analysis
cable 2357
geometric 2355, 2830, 2832
truss 2357
nonlinear cable analysis 2357
nonlinear cable members 801, 2332
nonlinear static analysis
verification problems 3583
nonlinear truss analysis 2357
nonlinearity
geometric 2381
material 2381
types 2381
nonsymmetric sections 1420
Norsok N-004 157, 158
Norwegian codes 2046
notional loads
adding 852, 3028, 3029
NRC
verification 3553
NRC 1995
seismic load definition 2548–2551
NRC 2005
seismic code 283
NS-EN 1993 1776, 1785, 1792
NS3473 1064
NS3473 Beam Design Brief 1331–1336
NS3473 Beam Design Principles 1065, 1066
NS3473 Column Design Brief 1337–1341
NS3473 Column Design Principles 1066
NTL file (AutoPIPE) 882
nuclear power plants 198

STAAD.Pro 5107 User Manual
NZS3404 1997
  design parameters 2032, 2037–2041
  material properties 2021
  member design 2025
  member property
  specifications 2021
  member resistance 2022
  output 2041, 2042
  section classification 2022

objects
  moving 699
  selecting 37
offsets 2964
OneSteel 204, 205
oneway loads
  square panel 2666, 2667, 2669, 2671
  towards option 291
open 2870
opening 31
opening a model
  bridge deck workflow 885
OpenSTAAD
  example files 4311
  functions 5057
optimization 1419, 2854
options
  setting 2413, 2414, 2419, 2422
orientation
  dialog 2936
  lamination 1539
  members 2503, 2505–2508
origin
  move, dialog 2903
  moving 700
orphan nodes 892
orthotropic material
  creating 791
orthotropic materials 2987
output
  ACI 318 beam design 1498–1500
  output width 2413
  overlapping plates 3097
  overview 2251

P
P-Delta
  KG 2797–2800
    large delta 2797–2800
    small delta 2797–2800
  stress stiffening 2797–2800
P-Delta analysis
  dynamic analysis 2351
  KG matrix 262
    large and small deltas 2349
    small delta 262
    stress stiffening 2350
page controls 1202
page eject 2423
page length 2423
page new 2423
pages
  Analytical Modeling 729
  Connections page 974
  Pushover page 3151
  RAM Connection mode 974
  Results 978
  Seismic Frame page 979
    See also page controls
panels
  pinning 2253
parameter specifications 2851, 2852
parametric mesh models 2434
parametric meshes
  add circular openings 659
  add density line 659
  add polygonal openings 658
  add surface model 657
parenthesis 2407, 2408
partial end releases 799
Partial Moment Release 2488–2490
pass through forces
  , See transfer forces
paste
  insertion point 2891
peformance checks 2392
perform rotation 2402
physical members
  modeling 314, 315
physical member
  rules for forming 1009
physical members
  assigning properties 315
  assigning sections 742
  assigning specifications 316, 804
  creating 315, 1009
  deleting 1009
design per NZS3404 2026–2028, 2030, 2031
  forming 1009
  forming, automatically 651
  forming, manually 650
general format 313, 314
loads 317
manually adding restraints 652
member end offsets 804
member query 245
releases 804
truss 804
physical modeler
  , See STAAD.Pro Physical Modeler
physical modeling
  dropping physical model 894
  load combinations 866
physical models
  examples 4311
picture album 1117, 1120
pictures
  copying 2250
  taking, tool 2247
pilecaps 4949
pinned supports
  assigning 812
pipe model
  support connection wizard 879
Pipe Model dialog 3111
pipe reactions
transfer to structure 881
pipe supports 301
Pipe Supports table 3117, 3118
PipeLink for STAAD.Pro 881
piping
  support connection wizard 180
piping mode
  model persistence 281
  supports 301
  updates 301
  See also piping workflow
Piping Mode
  connections, manually specifying 880
  import from AutoPIPE 878
  load data, transferring 881
  manual connections 880
  manual supports 880
  pages 883
  Pipe Model dialog 3111
  Support Connection Wizard 879
  supports, manual 880
  transfer load data 881
piping workflow 878
plan irregularities checking
  IS 1893 2016 87, 88
  plane frame structures 2409, 2410
  plane stress 2967
  plane stress elements 2498
  plastic hinges
    stiffness matrix 2392, 2393
  plate center stresses 3130, 3131
  plate element
    property, dialog 2981
  plate elements
    hydrostatic loading 833
    ignore inplane rotation 2967
    ignore stiffness 2968
    incidences 2431, 2432
    mesh generation 2433
    plane stress 2967
    pressure loads 829
    releases 2967
  rigid inplane rotation 2967
  plate girders
    AISC 9th Ed 1420
    plate mats 252
    plate node release 2967
    plate query 49
    plate specifications
      dialog 2967, 2968
    plates
      adding infill 655
      contour results 2217
      cut lines 2217
      drawing on nodes 654
      element loads 2657–2660
      ignore inplane rotation, assigning 809
      ignore stiffness 810
      incidences 2431, 2432
      inspect properties 49
      labels 617
      maximum number 2409, 2410
      modeling 653
      overlapping check 3097
      planar stress 808
      reference point 805
      reference point, dialog 2981
      releases, assigning 807
      results 2216
      rigid inplane rotation, assigning 809
      selecting 1146
      set attributes 653
      thickness, specifying 806
      warped 887
  plot settings
    changing 1215
  plotting
    copy picture 2250
    export view 2249
    print current view 2246
    take picture tool 2247
PMEMBER 313, 649
  See also physical members
pmembers
  load specifications 2655
PN EN 1993-1-1 1776, 1785, 1792
Poisson ratio 2503, 2505–2508
Polish NA
  Eurocode 3 160
post analysis plot 2849
post-analysis output 939
post-processing
  floor spectrum 199
post-tensioning
  force 2678, 2679
post-tensioning loads
  See poststress loads
postprocessing
  pages 2225
poststress loads 828
poststress member loads 2340
pre-analysis
  adding commands 936
pressure loads
  full plate 829
  partial plate 829
prestress loading
  example 4407, 4410, 4687, 4690
prestress loads 828, 2678, 2679
prestress member loads 2340
preview 1110–1114
primary elements 2388
primary load cases
  adding 822
  change load type 867
  define load type 867
print
  all 2840, 2841, 2844, 2845
  analysis results 2840, 2841, 2844, 2845
  asterisk meaning 2840, 2841, 2844, 2845
  cable sag 2840, 2841, 2844, 2845
cable sag 2840, 2841, 2844, 2845
cable sag 2840, 2841, 2844, 2845
diapragm cr 2840, 2841, 2844, 2845
diapragm cr 2840, 2841, 2844, 2845
displacements 2840, 2841, 2844, 2845
element forces 2840, 2841, 2844, 2845
element information 2840, 2841, 2844, 2845
element joint stress solid 2840, 2841, 2844, 2845
element stresses 2840, 2841, 2844, 2845
force envelope 2848, 2849
harmonic results 2815
joint coordinates 2840, 2841, 2844, 2845
material properties 2840, 2841, 2844, 2845
member forces 2840, 2841, 2844, 2845
member information 2840, 2841, 2844, 2845
member properties 2840, 2841, 2844, 2845
member stresses 2840, 2841, 2844, 2845
mode shapes 2840, 2841, 2844, 2845
section displacements 2846–2848
section forces 2840, 2841, 2844, 2845
steady state results 2815
story drift 2840, 2841, 2844, 2845
story stiffness 2840, 2841, 2844, 2845
support information 2840, 2841, 2844, 2845
support reactions 2840, 2841, 2844, 2845
surface forces 2846
Print Setup dialog 1107
print specifications 2840, 2841, 2844, 2845
printer plot 2849
printing
  center of rigidity 940
prismatic properties
dialog 2970–2972
tapered tubes 2467, 2468
prismatic tapered tubes 2467, 2468
problem initiation 2409, 2410
processor 28
program evaluated push load pattern 2823
project settings 4961
ProjectWise
  add 186, 192–194
  check in 2284
  Check In dialog 186, 192–194
  Integration Server 186, 192–194
  open from 2284
  sharing project to 2285
  Start Page 186, 192–194
  Update Server Copy 186, 192–194
ProjectWise Project
  registering 2277
ProjectWise projects
  assign project dialog 2275
  associating 2274
  registering 2277
ProjectWise Scenario Services
  , See Scenario Services
properties
  consolidating 249
dialog 2968
  elements 2487
  surface elements 2488
property reduction factors 2962
property specifications 2469, 2470
Publication T114 1407–1409
purlins
  Steel AutoDrafter 2235
push load pattern
  program evaluated 2823
user-defined 2823
pushover
defining 2387
example 5039
pushover analysis
beam results 2224
capacity curve 2223
defining 862
hinge results 2224
idealized capacity curve 2223
input parameters 2819
laoding 861
load steps 2223
nodal displacements 2224
objective 2377
overview 2376
purpose 2376
results 2223
scope in STAAD.Pro 2395
specifying 933
support reactions 2224
target displacement 2223
what is it 2376
pushover data
defining for members 862
pushover loading
adding 864
pushover spectral data 863
Q
QCE
calculation of 2383, 2384
quick access toolbar 57
R
RA angles
rotation 2301, 2302
RAM Concept 1353
RAM Connection
  CONNECT Edition 108
  CONNECT Edition V13 89
example 4925
multiple connections at a joint 232
RAM Connection mode
  about 953, 954
RAM Connection Mode
pages 974
settings 966
using 966
seismic frames
selection dialog 3175
seismic frames table 979
seismic load
adding load case 842
RPA (Algerian) 2546–2548
Seismic Load
Columbian 2571–2575
UBC 1997 2620–2623
seismic load definitions
IBC 2012 2596, 2598–2600
IBC 2015 2600, 2602
IBC 2018 2603–2605
NRC 1995 2548–2551
seismic load generator
UBC 2343
seismic loads
add structure weight 840
adding a definition 840
adding ASCE 7-10 definition 94, 115
adding ASCE 7-16 definition 87
IS 1893 2016 94
Japanese 2605–2607
NRC 2010 definition 2556, 2558, 2561
UBC 1985 2617, 2619, 2620
UBC 1994 2617, 2619, 2620
UBC 1997 2620–2623
use reference load case 841
seismic parameters
IS 1893 3047
seismic provisions
AISC 341 1388–1390
select members 2853, 2854
SELECT XM License 302
selecting
design groups 1148
members 1146
plates 1146
slabs 1146
selecting objects 37
selections
moving 699
selfweight
add load item 823
member lists 223, 224
separator 2407, 2408, 2422, 2423
SET command
CAXIAL 2413, 2414, 2419, 2422
settings
connection design 966
connection design dialog 977
SFS EN 1993-1-1 1776, 1785, 1792
sharing
e-mail project 51
to ProjectWise 51
shear
ACI 318 1497
shear area 2322
shear design
AISC LRFD 1454
shear links 1182
shear modulus 2503, 2505–2508
shear stress
AISC 9th Ed 1406
shear wall design 2862
shear walls 4952
shell elements
incidences 2431, 2432
shells
incidences 2431, 2432
shock spectrum 3627–3629
shortcuts 2268
shortcuts, keyboard 78
SI 2295
Singaporean National Annex 145
single angles
geometric axis 795
singly symmetric sections
AISC 9th Ed 1407
site class 2390, 2391, 2828
size specification 2849, 2850
slab design
starting 4953
slab design mode
entering 1215
slabs
construction drawings 4959
creating 1010
deleting 1010
design 4953, 4958
design calculations 4960
designing 1524
detailing 4959
forming 1010
rules for forming 1010
save project 4961
selecting 1146
toolbars 1200
uniform 1153
Slabs
1153, 1154
slender compression elements
AISC 9th Ed 1409
slenderness effects
ACI 318-14 1510
concrete design 1521, 1522
smart connections
dialog 3169
selecting 968
SNiP
wind load, parameters 3018, 3020
wind loads 837
wind parameters 3038, 3039
SNiP 2.01.07–85 203–205
SNiP 2.03.01-84 174
SNiP 2.23-81 162
SNiP II-7-81 2759, 2762, 2763
snippets
creating 2260–2262
editing 2260–2262
folders 2260–2262
inserting 2259
snow loads
ASCE 7-02 command 2785
snow loads
adding 851
ASCE 7-02 344–346, 851, 3021
definition 2641
soft story
spectra table 3027
spectral data
  defining 863, 3053
SRSS 2791–2795
SS EN 1993-1-1 1776, 1785, 1792
SSDD 1000
ST angles
  rotation 2301, 2302
STAAD Editor 2251
STAAD input files 49
STAAD input language
  conventions 2405
STAAD input language reference
  2404
STAAD PlanWin 895
STAAD SSDD 1000
STAAD.Pro
  about 28
  documentation 22–24
  modeling rules 2394
  scope of pushover analysis
    2395
  transferring loads to 887
Richardson et al. 2005
rail 974
railway
  2526–2528
  checking 132, 937
  IS 1893 2016 95, 96
railway track 2357
railway vehicle 2357
SPPM, See STAAD.Pro Physical Modeler
season 1771
seasons 1771
seasonal cycles 1771
seasonal factor 1771
seasonal loading 1771
seasonal variations 1771
seasons 1771
seismic design 297
seismic code 297
seismic design
  thesisme 297
seismic force
  loading 2650
  member 797
seismic loading
  definition 1771
seismic loads 297
seismic resistance
  loading 2650
  member 797
seismic response 297
seismic response
  loading 2650
  member 797
seismic study 297
seismic type
  loading 2650
  member 797
secondary moment 304
seeds
  check negative volume 893
  element loads 2660, 2661
  incidences 2432, 2433
  inspect properties 49
  Jacobian check 893
  maximum number 2409, 2410
solution
  eigenproblem 2363, 2364
  control defining 865, 2825, 3055
solvers
  advanced 267
  South African codes 2176
  SP 14.13330.2011 2764, 2766, 2767
  space bar 68
  space frame structures 2409, 2410
special selection of joints 3166
specifications
  loading 2650
  member 797
  member truss 2492, 2493
  member, adding 797
  nodes 810
  physical members 316
  plates 805, 2967, 2968
  whole structure 2959
specified sections
  member forces 2400
staed state
  examples 2817, 2818
  analysis end 2817
steady state analysis
  results printing 2815
Steel AutoDrafter
  create groups 2235
  draw a plan 2234
  drawing 2228–2231
  list 2232, 2233
  Drawing Style Manager 3160
  Grid Manager 3159, 3160
  groups list 2228–2231
  lists 2232, 2233
  opening workflow 2227
  systems of units 2228
steel connections
  add to report 972
  deleting 971
  designing 966
designing individually 971
ing editing 970
steel deck catalogs
ASC 695, 2918
VERCO 695, 2918
Vulcraft 695, 2918
steel design
AASHTO ASD 1464
AASHTO LRFD 1468
AASHTO specifications 1463
AIJ 2005 1976
AISC 360-10 verification problems 3916
AISC ASD verification problems 3988
AISC LRFD 1451
AISC LRFD verification problems 4034
American cold-formed 1473
ANSI/AISC N690-1984 1582
ANSI/AISC N690-1994 1575
API 1562
AS 4100 1633
AS 4100-1998 verification problems 3639
ASCE 10-97 1553
ASCE manuals and reports 1558
ASME NF 3000 1974 1588
ASME NF 3000 1977 1588
ASME NF 3000 1989 1597
ASME NF 3000 1998 1607
BS 5950-1-2000 verification problems 3787
BS 5950-5 1682
BS5400 1676
BS5950 1655
CAN/CSA-S16-09 1728
code checking 952
color results 331
commands 952
CSA CAN/CSA-S16-01 1701
designing members 952
DIN 18800 1880
Eurocode 1745, 1746, 1757, 1758
Eurocode 3 1754
Eurocode 3 DD ENV 1742
French code 1872
generating take off 953
initiating 951
IS 800 1916
IS 800 2007 1942
IS801 1930
NORSOK N-004 2095
NS 3472 NPD 2046
NTC 1987 2013
NZS 3404 1997 2021
S136-94 1715
SAB0162-1 1993 2181
SANS 10162-1-2011 2200
selecting members 952
SNiP 2.23-81 2137
SP 16.133330.2011 2152
specifications 2850
specifying parameters 951
UK example 1 4627, 4632, 4633
UK example 2 4645, 4649
US example 1 4313, 4317, 4318
US example 2 4363, 4366, 4367
verification examples 3639
web openings 372, 1432
steel joists
adding sections 731, 732
sizing 731, 732
steel moment frames 2386
steel section libraries
AISI sections 1474
steel section library
Australian 1635–1638
British 1657–1659
Canadian 1701–1704
French 1877–1879
German 1881–1883
Russian 2138, 2139, 2154
South African 2182–2185
steel sections
APL Apollo 91
built-in libraries 1368, 2325, 2461, 2464, 2465
steel table 2969
stiffness
ignore for plates 2968
stiffness analysis 2346–2348
stiffness matrix
reseting 2835, 2836
with plastic hinge 2392, 2393
storage 28
story drift
checking 133, 941
IS 1893 2721, 2726, 2727, 2729, 2730, 2732, 2737, 2739–2743
printing 2840, 2841, 2844, 2845
story stiffness
printing 2840, 2841, 2844, 2845
STP file
create 194, 196, 197
strain loads 2341, 2682
strength design
ACI 318-14 1496
stress stiffening 2350
stress stiffening matrix 2797–2800
stress/force output 2846
stretch members 360, 364
strings 2258
Struclink
member attributes 2535, 2536
Structural Synchronizer 2269, 2270
structure checks
overlapping plates 3097
structure colors 1132
structure geometry
rotation 2444, 2445
Structure View window 1207, 1209
structure wizard
trusses 712
units 711
structures
  check for multiple 887
types 2294
subgrade modulus
  spring supports 2517–2519
substitute 2402
support connection wizard 180
Support Connection Wizard 879
support displacement loads
  2342, 2683, 2684
support displacements
  enforced supports 813
support information
  printing 2840, 2841, 2844, 2845
support reactions
  printing 2840, 2841, 2844, 2845
support settlement
  UK example 5 4683–4685
  US example 5 4403–4405
supports
  compression-only 2986
  compression-only springs 816
create dialog 2983–2986
custom releases 814
displacement 825
enforced 813, 2984
enforced but 813, 2984
fixed 812, 2983
fixed but 814, 2983
foundations 818, 2985
inclined 817, 2516, 2517, 2984
information, printing 2840, 2841, 2844, 2845
multilinear springs 2985
multilinear springs 815, 2520, 2521
pinned 812, 2983
piping, manually drawing 880
reaction labels 2213
specifications 2513
specifying 2514, 2515
springs, linear 814
tension-only 2986
tension-only springs 816
whole structure dialog 2982
surface constants 2508, 2509
surface elements
  constants 2508, 2509
  incidences 2433
  loads 2663, 2664
  properties 2488
  stress/force output 2846
surface loads 2663, 2664
surface mesh
  starting 661
  technical notes 666, 667
surface query 49
surfaces
  add self weight 832
  incidences 2433
  inspect properties 49
  openings:commands:SURFACE INCIDENCES 2433
  properties 2487
system of units
  changing 40
system requirements 28

T

  table rows
    using 1119
tables
  dialog 2937
  Pipe Supports 3117, 3118
tabulated results
  AISC LRFD 1458
take off specification 2856, 2857
take picture tool 2247
Tandem 3138
tapered
  I section, assigning 734
tapered I-shape
  dialog 2972
tapered members
  BS 5950-2000 1675, 1676
  tapered sections 2326, 2468, 2469
tapered tubes 2467, 2468
Technical Reference Manual 27, 28
technical support 81, 82
temperature loading
  verification problem, beam 3231–3233
  verification problem, plate 3388–3390, 3392, 3395
temperature loads 2341, 2682
tension stress
  AISC 9th Ed 1406
tension-only members
  specifying 798
tension-only springs 2522–2524
tension-only springs 251
tension-only supports 2986
term 288, 293, 295, 300, 306–308, 310, 312, 321, 322, 332–334, 336, 4949, 4950, 4957, 4970
terrain 3161, 3162
text
  insert to view 630
thermal expansion, coefficient
  2503, 2505–2508
thickness
  element, dialog 2981
timber
  design codes 1364
timber design
  AITC 1526
  CAN/CSA-086-01 1720
code checking 1365, 2858
codes 2857
commands 1365
design parameters 2857
designing members 1365
Eurocode 5 1856
initiating 1365
member selection 2858, 2859
selecting members 1365
specifications 2857
specifying parameters 1365
verification examples 4296
timber section library
  AITC 1532, 1533
  Canadian 1721, 1722
timber table 2969
time history
  floor spectra 935
time history analysis 2373, 2374
time history load definitions 3056, 3057
time history loading
  verification problem 3627–3629
time history loads
  adding load 861
  define by data pairs 855
  define by function 856
  define by spectrum 857
  define with external data 859
  defining parameters 860
  floor spectrum 2833, 2834
  generate spectrum output 858
  use frequency-spectra pairs 859
time history response 2373, 2374
time varying loads
  response history analysis 2767–2769
tool tips
  structural 620, 621
toolbars
  fly-out 244
  Rotation 636
torsion
  ACI 318 1497
  Eurocode 3 158, 159
  FIXED BUT supports 1407–1409
  publication T114
  1407–1409
torsion design
  AISC 360-16 97
torsional irregularities 2531
transfer forces
  reporting 328, 329
  Transfer Pipe Reactions to Structural Model dialog 3119
  translate
    copies 702
    model 700
  translational repeat 2892
  transmission tower codes
    ASCE 10-97 1553
    triangulation 3137
  truss members
    compression-only 2495–2497
    specifying 798
    tension-only 2495–2497
  truss structures 2409, 2410
  trusses
    structure wizard 712
  TS 500 1078
  TS 500 Beam Design Brief 1078–1080
  TS 500 Beam Design Principles 1078–1080
  TS 500 Column Design Principles 1080
  TS-500 1078
  TS500 Beam Design Brief 1347, 1349
tutorial
  Advanced Concrete Design workflow 4949
  macro 5060
tutorials
  reinforced concrete frame 449
  slab analysis 535
  steel portal frame 373
types of nonlinearity 2381
  types of structures 2294
typographical conventions 2407, 2408

UBC 1985 seismic load 2617, 2619, 2620
UBC 1994 seismic load 2617, 2619, 2620
UBC 1997 seismic load 2620–2623
UBC seismic load generator 2343
UK NA
  Eurocode 3 178
UK steel design
  verification 3787
underlining 2407, 2408
undo 1117, 2882
uniform loads 826
uniform slab 1153
unit conversion
  utility 43, 44
unit converter
  section wizard 785, 786
units
  change input 41
  current 39, 40
  display 39, 40
  force 2411–2413
  in dialogs 42
  length 2411–2413
  structure wizard 711
  switching 2411–2413
  systems of 2295
unsymmetric sections 1420
updates
  product version 2283
UPT
  export from Section Wizard 769
US concrete design
  verification 4250
US steel design
  verification 3828
user interface
  overview 21
user provided table
  select section type dialog 2975
user provided tables
  create, dialog 2974
user steel tables 2446–2448
user tables
general 355, 356
user tools
  add macros 5075
  running 5052
user-defined push load pattern 2823
user-defined vehicles 3138
user-provided steel tables 2325, 2446–2448, 2458, 2469, 2470
user-provided table
  from Section Wizard 739
user-provided tables
  create general section 735, 739
dialog 2973
wide flange 2976
using the bridge deck mode 884
utilities
  calculator 3101
  unit converter 43, 44
utilization ratio
  display for steel design 2216

\[ V \]
validating 887
vehicle loads
  defining 853
vehicles
  custom 3138
  database 3138
  display colors 3137
  user-defined 3138
VERCO steel decks 695, 2918
verification problem
  AASHTO ASD 3828–3831, 3833, 3835–3839
  AASHTO LRFD 3840–3842, 3844, 3845
  AIJ 1984–1986
  ASME NF 3000 1974 1595
  British Cold Formed Steel 1690, 1693, 1694
  CAN-CSA 086-01
  4296–4298
  CAN-CSA S16-01
  3650–3663
  CAN-CSA S16-14
  AIJ 2002 Check for MISES parameter
  3771–3774
  AISC 360-10 steel design 3916
  AISC ASD steel design 3988
  AISC LRFD steel design 4034
  AS 4100-1998 steel design 3639
  ASME NF 3000 2004 STYPE 1 Pipe 4242, 4244–4246
  beams 3195
  BS 5950-1-2000 steel design 3787
  CSA S16-09 - Axial Tension 3664–3666
  CSA S16-09 - Beam Bending 3669–3671
  CSA S16-09 - Beam Shear Capacity 3673–3675
  CSA S16-09 - Select a Beam 3679–3681
  CSA S16-09 - Shear Capacity Combined Stresses 3687–3690
  Eurocode 5 1866–1872
  SAB0162-1 2193–2199
  ACI 318 concrete design
  4250
  AIJ 2002 Check for MBG parameter
  3766–3770
  AIJ 2002 Check for MISES parameter
  3771–3774
  AISC 360-10 steel design 3916
  AISC ASD steel design 3988
  AISC LRFD steel design 4034
  AS 4100-1998 steel design 3639
  ASME NF 3000 2004 STYPE 1 Pipe 4242, 4244–4246
  beams 3195
  BS 5950-1-2000 steel design 3787
  CSA S16-09 - Axial Tension 3664–3666
  CSA S16-09 - Beam Bending 3669–3671
  CSA S16-09 - Beam Shear Capacity 3673–3675
  CSA S16-09 - Select a Beam 3679–3681
  CSA S16-09 - Shear Capacity Combined Stresses 3687–3690
  Eurocode 5 1866–1872
  SAB0162-1 2193–2199
  ACI 318 concrete design
  4250
  AIJ 2002 Check for MBG parameter
  3766–3770
  AIJ 2002 Check for MISES parameter
  3771–3774
  AISC 360-10 steel design 3916
  AISC ASD steel design 3988
  AISC LRFD steel design 4034
  AS 4100-1998 steel design 3639
  ASME NF 3000 2004 STYPE 1 Pipe 4242, 4244–4246
  beams 3195
  BS 5950-1-2000 steel design 3787
  CSA S16-09 - Axial Tension 3664–3666
  CSA S16-09 - Beam Bending 3669–3671
  CSA S16-09 - Beam Shear Capacity 3673–3675
  CSA S16-09 - Select a Beam 3679–3681
  CSA S16-09 - Shear Capacity Combined Stresses 3687–3690

VERCO steel decks 695, 2918
verification problem
  AASHTO ASD 3828–3831, 3833, 3835–3839
  AASHTO LRFD 3840–3842, 3844, 3845
  AIJ 1984–1986
  ASME NF 3000 1974 1595
  British Cold Formed Steel 1690, 1693, 1694
  CAN-CSA 086-01
  4296–4298
  CAN-CSA S16-01
  3650–3663
  CAN-CSA S16-14
  AIJ 2002 Check for MISES parameter
  3771–3774

natural frequencies 3611–3620
natural modes 3621–3625

AASHTO LRFD 3840–3842, 3844, 3845
AIJ 2005 1984–1986
AS 4100-1998 steel design 3639
ASME NF 3000 2004 STYPE 1 Pipe 4242, 4244–4246
beams 3195
BS 5950-1-2000 steel design 3787
CSA S16-09 - Axial Tension 3664–3666
CSA S16-09 - Beam Bending 3669–3671
CSA S16-09 - Beam Shear Capacity 3673–3675
CSA S16-09 - Select a Beam 3679–3681
CSA S16-09 - Shear Capacity Combined Stresses 3687–3690
CSA S16-09 - Short Column Compression 3694–3696
CSA S16-09 - Slender Column Compression 3700–3702
CSA S16-09 - Wide Flange Capacity Combined Stresses 3706–3709
dynamic analysis 3592
frames 3269
general analysis 3422
IBC 2018 Static Seismic T 1.2 3440, 3443, 3445–3447
IBC 2018 Static Seismic T Greater Than 2.5 3450, 3451, 3454, 3456, 3460
IBC 2018 Static Seismic T Less Than 0.5 3468, 3471, 3473–3475
IS 1893 2016 GL Calculation 3497, 3499, 3501, 3502
IS 1893 2016 Irregular Modes of Oscillation 3509–3512
IS 1893 2016 Mass Irregularity 3519–3522
IS 1893 2016 Re entrant Corners 3530–3533
IS 801-Beam with axial and major axis bending 3753, 3757–3759
IS 801-Column with axial and major axis bending 3761, 3763, 3764
IS13920 2016 Rectangular Beam 4281–4285, 4287, 4289–4294
natural frequencies 3611–3620
natural modes 3621–3625
natural modes, beam 3608–3610
nonlinear static analysis 3583
NZS3404 1997-UB Section 3776, 3777, 3779, 3782
time history loading 3627–3629
trusses 3244
Wind On Open Structure 3579, 3580, 3582, 3583
vertical axis
set Z up 46
vibration analysis 365, 368–370
video files 2213
view window
quick commands menu 68
right-click pop-up menu 61, 63
right-click view tools 64
views
adding to 1135
creating new 627, 1015
cutting sections 623
exporting 2249
open existing 1134, 1135
portions 623
print current 2246
renaming 1136
reopening saved 1016
save as 1136
saving 1016
Vulcraft steel decks 695, 2918

W
wall-slab meshing 348
walls
adding 4952
warped plates
checking 887
tolerance 2938–2941
web openeings
AISC ASD 372
designing beams with 372
web openings
designing beams with 1432
web tapered sections
design 1377, 1409
welcome 21
welded plate girders 1371
Whats New? 84
wide flange
user-defined 2976
width
input 2413
output 2413
wind
ASCE 7 wind loading 836
wind intensity 3038, 3039
wind load generators 2345
wind loads
applying 838
ASCE 7 3039–3042
ASCE 7-02 337, 339, 341, 343
ASCE 7-2010 intensity 140
definition 2623–2626, 2628, 2629
dynamic per SP 20 839
exposure 2623–2626, 2628, 2629
Russian 203–205
SNiP 205, 3018, 3020
SNiP 85 (Russian) 2623–2626, 2628, 2629
SNiP parameters 3018, 3020
SP 20.13330.2011 (Russian) 2623–2626, 2628, 2629
wind pressure 3038, 3039
Windows
supported versions 28
Wood and Armer
moments 1682
wood design 1526
workflows
building planner 895
Foundation Design 1359
physical modeling 894
write text 630

X
XML
connection tags schema 986, 988–993

Y
Young’s modulus 2503, 2505–2508

Z
Z up
configuring 323
zero length members 890
zoom area 1121
zoom extents 1218